WL-TR-93-3047

COMPUTATIONAL FLUID DYNAMICS (CFD) RESEARCH BRANCH TECHNICAL BRIEFS

Don W. Kinsey et al.

15 May 1993

Final Report for Period May 1992 - May 1993

Approved for public release; distribution is unlimited.

FLIGHT DYNAMICS DIRECTORATE
WRIGHT LABORATORY
AIR FORCE MATERIEL COMMAND
WRIGHT-PATTERSON AIR FORCE BASE, OHIO 45433-7562

NOTICE

When Government drawings, specifications, or other data are used for any purpose other than in connection with a definitely Government-related procurement, the United States Government incurs no responsibility or any obligation whatsoever. The fact that the government may have formulated or in any way supplied the said drawings, specifications, or other data, is not to be regarded by implication, or otherwise in any manner construed, as licensing the holder, or any other person or corporation; or as conveying any rights or permission to manufacture, use, or sell any patented invention that may in any way be related thereto.

This report is releasable to the National Technical Information Service (NTIS). At NTIS, it will be available to the general public, including foreign nations.

This technical report has been reviewed and is approved for publication.

Don W. Kinsey

DON W. KINSEY, Tech Mgr
Interdisciplinary and Applied CFD Section

J. M. Manter

JOSEPH M. MANTER
Chief
CFD Research Branch

Dennis Sedlock

DENNIS SEDLOCK
Acting Chief
Aeromechanics Division

If your address has changed, if you wish to be removed from our mailing list, or if the addressee is no longer employed by your organization please notify WL/FIMC, WPAFB, OH 45433-7562 to help us maintain a current mailing list.

Copies of this report should not be returned unless return is required by security considerations, contractual obligations, or notice on a specific document.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No 0704-0188

1. AGENCY USE ONLY *(Leave blank)*	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	9 Jun 93	Final Report May 92 - May 93

4. TITLE AND SUBTITLE
Computational Fluid Dynamics Research Branch Technical Briefs

5. FUNDING NUMBERS
PE: 62201F
PR: 2404
TA: 10
WU: A1

6. AUTHOR(S)
Don W. Kinsey, et al.

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Flight Dynamics Directorate
Wright Laboratory
Air Force Materiel Command
Wright-Patterson AFB OH 45433-7562

8. PERFORMING ORGANIZATION REPORT NUMBER
WL-TR-93-3047

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
Flight Dynamics Directorate
Wright Laboratory
Air Force Materiel Command
Wright-Patterson AFB OH 45433-7562

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
WL-TR-93-3047

11. SUPPLEMENTARY NOTES
Technical Summaries of in-house work completed during the previous year. Work covers both basic research (6.1) and research and development (6.2).

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution unlimited.

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*
This report consists of brief technical descriptions and one or more pictures of charts depicting research conducted in-house during the past year. The briefs are designed to provide enough information to clearly define the task, but are short enough to hold the reader's interest. Where applicable, a reference for more detailed information is provided. The research covers a wide spectrum of work; from grid generation to flow solvers, to post-processors; and from incompressible to hypersonic speeds.

14. SUBJECT TERMS
Computational Fluid Dynamics, Grid Generation, Hypersonic Flow, Unstructured Meshes.

15. NUMBER OF PAGES
31

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	UL

GENERAL INSTRUCTIONS FOR COMPLETING SF 298

The Report Documentation Page (RDP) is used in announcing and cataloging reports. It is important that this information be consistent with the rest of the report, particularly the cover and title page. Instructions for filling in each block of the form follow. It is important to ***stay within the lines*** to meet ***optical scanning requirements***.

Block 1. Agency Use Only *(Leave blank)*

Block 2. Report Date. Full publication date including day, month, and year, if available (e.g. 1 Jan 88). Must cite at least the year.

Block 3. Type of Report and Dates Covered. State whether report is interim, final, etc. If applicable, enter inclusive report dates (e.g. 10 Jun 87 - 30 Jun 88).

Block 4. Title and Subtitle. A title is taken from the part of the report that provides the most meaningful and complete information. When a report is prepared in more than one volume, repeat the primary title, add volume number, and include subtitle for the specific volume. On classified documents enter the title classification in parentheses.

Block 5. Funding Numbers. To include contract and grant numbers; may include program element number(s), project number(s), task number(s), and work unit number(s). Use the following labels:

C	- Contract	**PR**	- Project
G	- Grant	**TA**	- Task
PE	- Program Element	**WU**	- Work Unit Accession No.

Block 6. Author(s). Name(s) of person(s) responsible for writing the report, performing the research, or credited with the content of the report. If editor or compiler, this should follow the name(s).

Block 7. Performing Organization Name(s) and Address(es). Self-explanatory.

Block 8. Performing Organization Report Number. Enter the unique alphanumeric report number(s) assigned by the organization performing the report.

Block 9. Sponsoring/Monitoring Agency Name(s) and Address(es). Self-explanatory.

Block 10. Sponsoring/Monitoring Agency Report Number. *(If known)*

Block 11. Supplementary Notes. Enter information not included elsewhere such as: Prepared in cooperation with...; Trans. of...; To be published in.... When a report is revised, include a statement whether the new report supersedes or supplements the older report.

Block 12a. Distribution/Availability Statement. Denotes public availability or limitations. Cite any availability to the public. Enter additional limitations or special markings in all capitals (e.g. NOFORN, REL, ITAR).

DOD - See DoDD 5230.24, "Distribution Statements on Technical Documents."
DOE - See authorities.
NASA - See Handbook NHB 2200.2.
NTIS - Leave blank.

Block 12b. Distribution Code.

DOD - Leave blank.
DOE - Enter DOE distribution categories from the Standard Distribution for Unclassified Scientific and Technical Reports.
NASA - Leave blank.
NTIS - Leave blank.

Block 13. Abstract. Include a brief ***(Maximum 200 words)*** factual summary of the most significant information contained in the report.

Block 14. Subject Terms. Keywords or phrases identifying major subjects in the report.

Block 15. Number of Pages. Enter the total number of pages.

Block 16. Price Code. Enter appropriate price code *(NTIS only)*.

Blocks 17. - 19. Security Classifications. Self-explanatory. Enter U.S. Security Classification in accordance with U.S. Security Regulations (i.e., UNCLASSIFIED). If form contains classified information, stamp classification on the top and bottom of the page.

Block 20. Limitation of Abstract. This block must be completed to assign a limitation to the abstract. Enter either UL (unlimited) or SAR (same as report). An entry in this block is necessary if the abstract is to be limited. If blank, the abstract is assumed to be unlimited.

Contents

DTIC QUALITY INSPECTED 3

Introduction

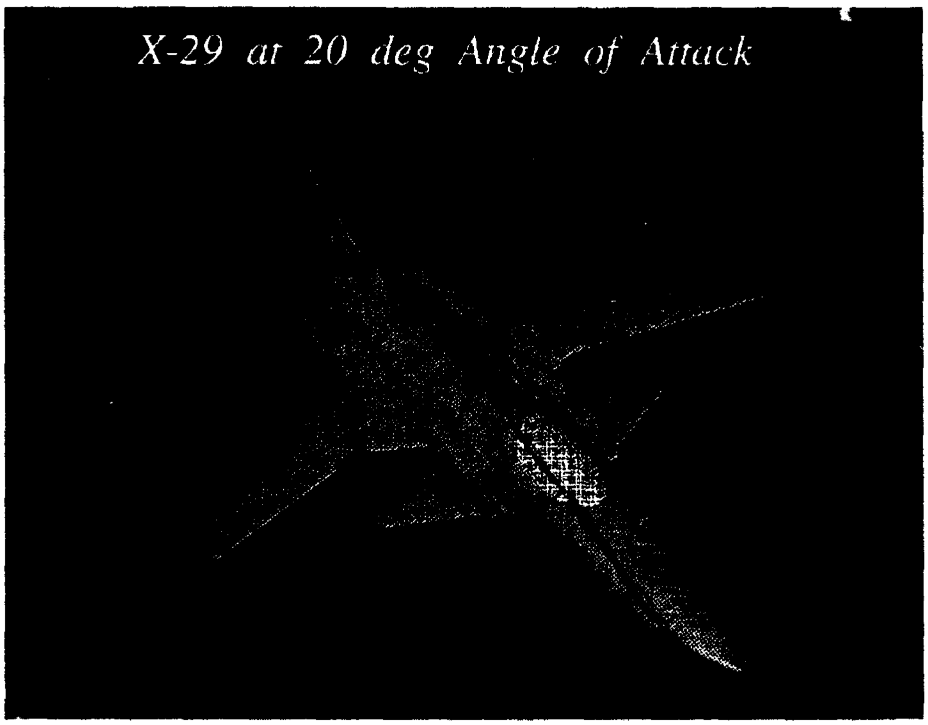
Don W. Kinsey
CFD Research Branch

This technical report was produced by the Computational Fluid Dynamics Branch of the Aeromechanics Division, Flight Dynamics Directorate, Wright Laboratory. The CFD Branch is composed of two Sections; the CFD Research Section, and the Interdisciplinary and Applied CFD Section. The CFD Research Section conducts basic research which is largely sponsored by the Air Force Office of Scientific Research. The IACFD Section conducts research and development of a more applied nature. Thus the CFD Branch as a whole covers a very wide range of technologies from geometry generation to post-processing of CFD results.

CFD technology is expanding very rapidly. New algorithms, techniques and applications are being added almost daily. Some of these represent significant advances in the state-of-the-art, while others are of value to a more limited audience. Most of the more complete results are published in symposium papers and journal articles, but many smaller efforts are never published. In either case it is easy for the work to become lost; the published work is lost in the volumes of papers and articles that we never seem to have time to read, and the unpublished works are lost in the files of the researcher. The objective of this report is to document in a single source the preceding year's CFD accomplishments.

The technical descriptions provided in this report follow closely the format used in NASA's National Aerodynamic Simulation annual report. The articles are designed to provide enough information to clearly define the objective, approach and significant accomplishments of the task, but be short enough to hold the reader's interest. Where applicable, a reference that contains more detailed information is provided. In all cases, the name of the author is provided and he may be contacted directly for details about his work. An alphabetic listing of each author, his phone number, mailing address and e-mail address is provided on the last of the report.

The picture chosen to go with this introduction is representative of our current capability in applied CFD. The geometry was digitized from blueprints, the different parts of the geometry were combined, and the surface grid and block faces were produced using our I3G/VIRGO interactive graphics package. The volume (3-D) grid was computed with a code called PLUTO. An Euler solver called MERCURY, based on Jameson's finite volume technique, produced the flow solution. The surface geometry and particle traces were produced with the NASA-developed PLOT3D graphics package.



Euler Solution on a Complex Fighter Geometry

Three-Dimensional Unstructured Grid Generation

Frank C. Witzeman Jr
Interdisciplinary and Applied CFD Section

Research Objective

One of the most difficult and time-consuming tasks in the CFD solution process is grid generation. Interactive graphics-based programs such as GRIDGEN and I3G/VIRGO have overcome many of the limitations and labor-intensive tasks of multiple-block structured grid generation. Although these methods (and many others) have enabled CFD solution turnaround times on the order of weeks or days, they cannot meet design environment goals of days or hours. This effort is focused on reducing CFD solution times from weeks to days through the development of a highly-automated, easy-to-use unstructured grid generation procedure. This method will be applicable to arbitrary and complex vehicles and components which are nearly impossible to model with the structured grid methods.

Approach

The unstructured grid generation method is part of the overall Multi-Bodied CFD Method being developed under contract to the Computational Mechanics Company (Dr S. R. Kennon, Principal Investigator). The method is based on a generalized three-dimensional Delaunay concept which produces grids comprised of tetrahedral elements instead of structured, hexahedral elements. For most cases, the only input required is an accurate description of the geometry. Currently, this geometry description is the same as that used for the structured-grid methods. Unique features of this method are the automatic addition of grid points (both on- and off-body) and the enforcement of triangular faces which lie on the geometric surfaces. Further advantages of the method include the automatic insertion of grid points in regions of interest (adaptive gridding) and relocation of grid points for moving-body simulations.

Accomplishments

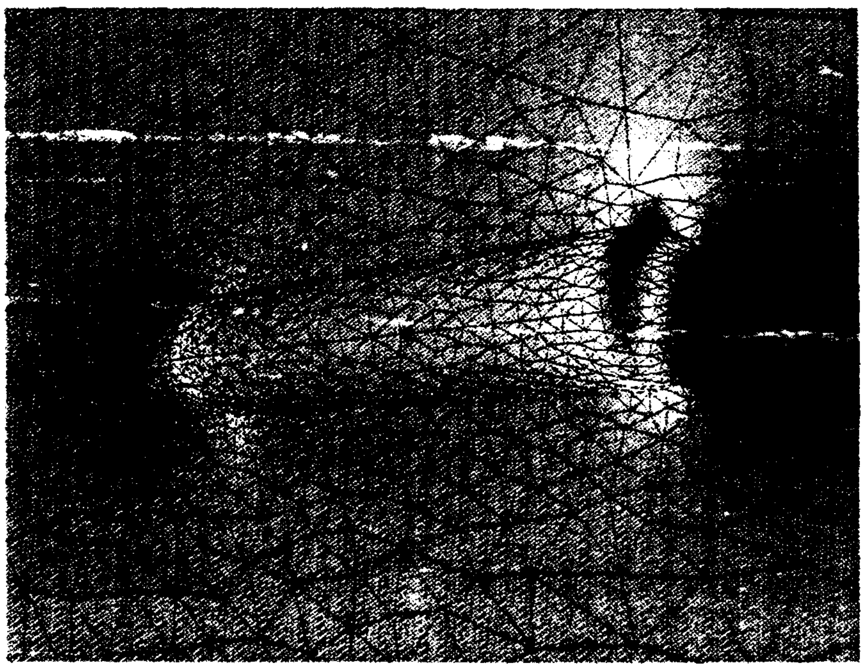
A preliminary version of the unstructured grid generator, TETMESH, has been produced. The software is being evaluated by several Air Force organizations. Three-dimensional unstructured grids have been constructed for several test cases, and Euler solutions have been obtained using our in-house flow solver COBALT. For the example illustrated below, approximately 1 week was required to generate a multiple-block structured grid. For the same configuration, only 6 hours were needed to generate the unstructured grid.

Future Plans

The interactive three-dimensional unstructured grid generator will be modified to be more robust and user-friendly. More generalized criteria will also be developed for unstructured grids made up of irregular tetrahedral and triangular-prismatic elements. These types of grids are required for accurate simulation of viscous flows.

Publications

Kennon, S. R., et al. "Geometry Based Delaunay Tetrahedralization and Mesh Movement Strategies for Multi-Body CFD." AIAA Paper 92-4575, August 1992.



Unstructured Grid and Pressure Contours on a Decoy Body and Planar Slice

Navier-Stokes Simulation on Unstructured Hexahedral Meshes

Captain Michael J. Aftosmis
CFD Research Section

Research Objective

This research effort develops and evaluates an adaptive, high-resolution, upwind algorithm for the simulation of inviscid and viscid flows. Upwind methods provide discrete representations of shocks and contact discontinuities using as few cells as possible, and adaptive mesh techniques automatically adjust the local flow physics. The combination of these two approaches suggests a natural and extremely flexible platform for simulating flowfields with features spanning a variety of disparate length scales. This work seeks first to overcome some of the practical and implementational issues involved with the development of such a technique, and then to evaluate the utility of the resulting method in its application to a variety of problems of interest.

Approach

This effort applies a Roe-based, finite-volume, upwind technique on an adaptive mesh consisting of unstructured hexahedral cells. Viscous terms are modeled with compact central differences. As the solution evolves, the mesh cells adapt through directional cell division in order to increase the resolution of the computation in and around emerging flow features. By concentrating new computational nodes only where the solution requires further enhancement, the method uses far fewer cells than comparable structured mesh techniques.

Accomplishments

An unstructured computer code was written and validated on a variety of inviscid and viscous test cases. Numerical simulations of 3D supersonic corner flows, sub- and transonic airfoil flows, and simulation of laminar flow over a delta wing were all in agreement with experimental and theoretical results. The method typically requires 10 to 20 times fewer cells than an equivalently resolved solution on a structured mesh. Moreover, since the number of operations which must be performed at each time step is proportional to the number of cells, each iteration requires less computer time. Thus, overall turnaround time is greatly reduced.

Future Plans

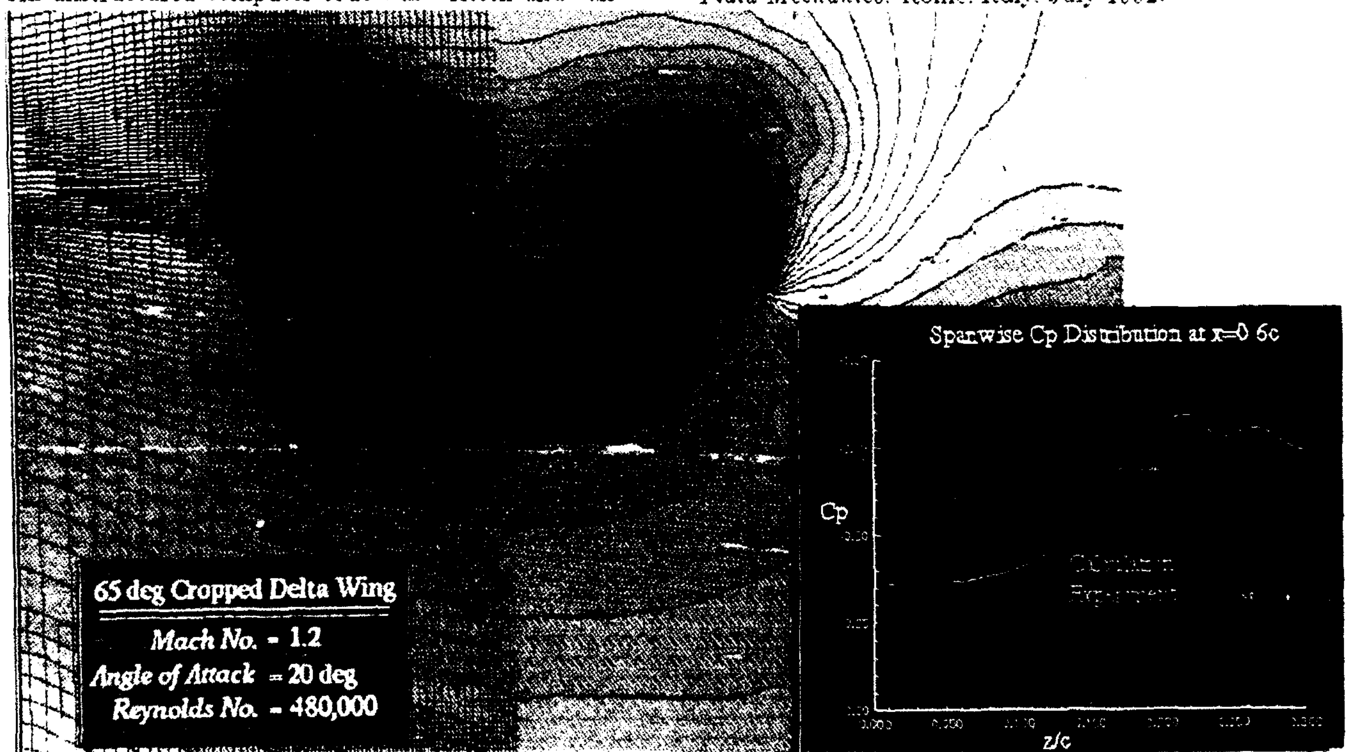
The method will be applied to time-dependent problems including those containing significant geometric complexity. Evaluation of the performance of the technique for such problems is essential before the technique will be considered mature.

Publications

Aftosmis, M.J., "An Upwind Method for Simulation of Viscous Flow on Adaptively Refined Meshes," Accepted for publication by the *AIAA Journal*, Mar. 1993.

Aftosmis, M.J., "Viscous Flow Simulation Using and Upwind Method for Hexahedral Based Adaptive Meshes," AIAA Paper 93-0772, Jan. 1993.

Aftosmis, M.J., "A Second-Order TVD Method for the Solution of the 3D Euler and Navier-Stokes Equations on Adaptively Refined Meshes," *Proceedings of the 13th International Conference on Numerical Methods in Fluid Mechanics*, Rome, Italy, July 1992.



Pressure contours for viscous supersonic flow over cropped delta wing. The final adapted mesh contains 999,000 nodes. The inset shows a comparison with wind tunnel data.

COBALT - An Unstructured Flow Solver

Willam Z Strang
Interdisciplinary and Applied CFD Section

Research Objective

To simplify computational fluid dynamics (CFD) analyses of air vehicles of arbitrary geometrical complexity: to lessen the time required to generate such predictions by a factor of 5: to reduce the CFD skill level required of the engineers conducting the analyses.

Approach

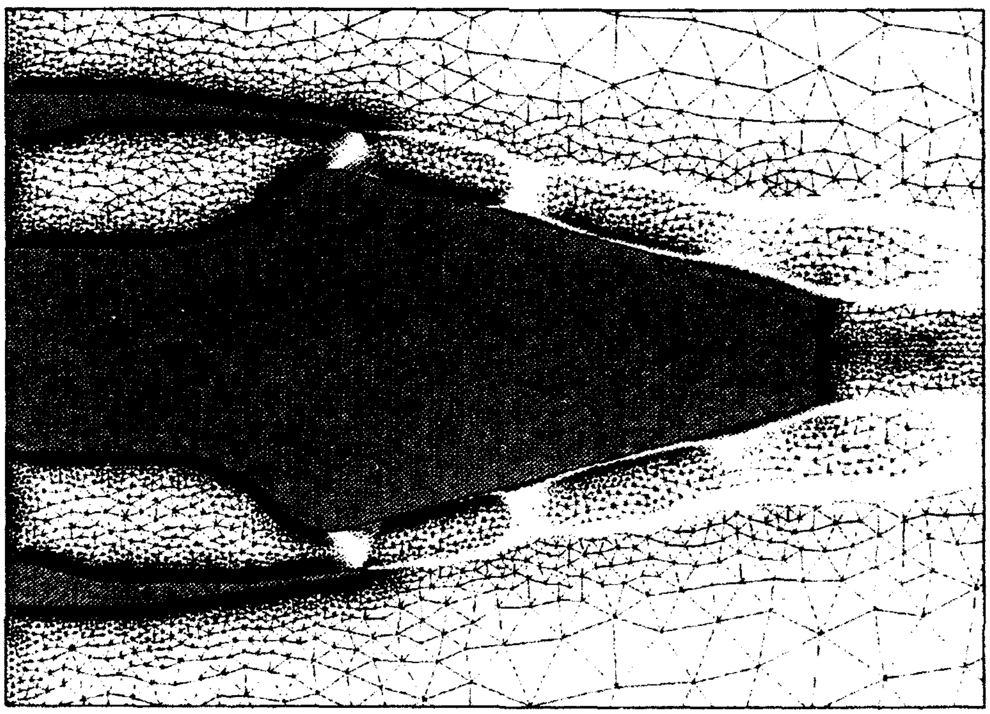
Unstructured grids are one of the very few methods, if not the only method, offering the potential to achieve the above research objectives. In theory, a user generates only the bounding surfaces of the problem and the unstructured mesh generator automatically tessellates the domain interior. Unstructured grids, however, require flow solvers that are capable of handling general connectivity issues. Thus, an in-house project to develop an unstructured flow solver, COBALT, began in February 1990.

Accomplishments

The fundamental algorithm of COBALT is the spatially and temporally first-order accurate, non-linear scheme due to Godunov. The extension to second-order accuracy in space is due to van Leer and Collela. Second- and third- order accuracy in time is achieved with classical Runge-Kutta integration schemes. Monotonicity, required for stability of the spatially second-order scheme, is based on the works of Collela, Goorjian and Obayashi. Slip wall, adiabatic and isothermal *no-slip wall*, *farfield*, *inflow* and *outflow* boundary conditions are available. Considerable effort has been devoted to these boundary conditions to make the code as robust and as accurate as possible. Construction of the viscous terms follows MacCormack's work. The Baldwin and Barth turbulence model is used to simulate the effects of fine scale turbulence. Two-dimensional, axi-symmetric and three-dimensional problems may be treated. The sole requirement of the mesh is that it be composed of convex cells. Lastly, with the exception of a few pre- and post-processing tasks, COBALT is fully vectorized and typically operates in excess of 100 MFLOPS a Cray X-MP/216.

Future Plans

Additional validation of the viscous and turbulence model routines, documentation, third-order spatial accuracy, parallel implementation, additional turbulence models and grid adaptation are planned.



Turbulent Solution of an Axi-Symmetric Plug Nozzle

Unstructured Approach to the Design of Multiple-Element Airfoils

Don W. Kinsey
Interdisciplinary and Applied CFD Section

Research Objective

Development of efficient high-lift airfoils may lead to reduced weight and complexity over current high lift systems with attendant reduced cost and improved reliability. The research described here combines the relatively new unstructured grid computational fluid dynamics (CFD) technique with an existing inverse design procedure. The results will, hopefully, be a new technique to design and analyze high-lift airfoils with the improved accuracy of an Euler solver, but significantly less work for the designer than most advanced CFD methods.

Approach

The CFD solver was an implicit Van Leer split flux Euler scheme with point Gauss-Seidel relaxation for two-dimensional triangular grids. A two-dimensional unstructured grid generator based on the Delaunay triangulation process was used to create the grid. Grid movement, needed to allow the geometry to change with the inverse design procedure, was provided by a "spring analogy" process. The inverse design program is a variation of the Modified Garabedian-McFadden (MGM) residual-correction procedure. This procedure requires a target velocity distribution and an initial airfoil geometry. Then, through a series of iterations, the airfoil geometry is adjusted based on the difference between the current and target velocity distributions. The converged solution provides a geometry that produces the target velocity distribution.

Accomplishments

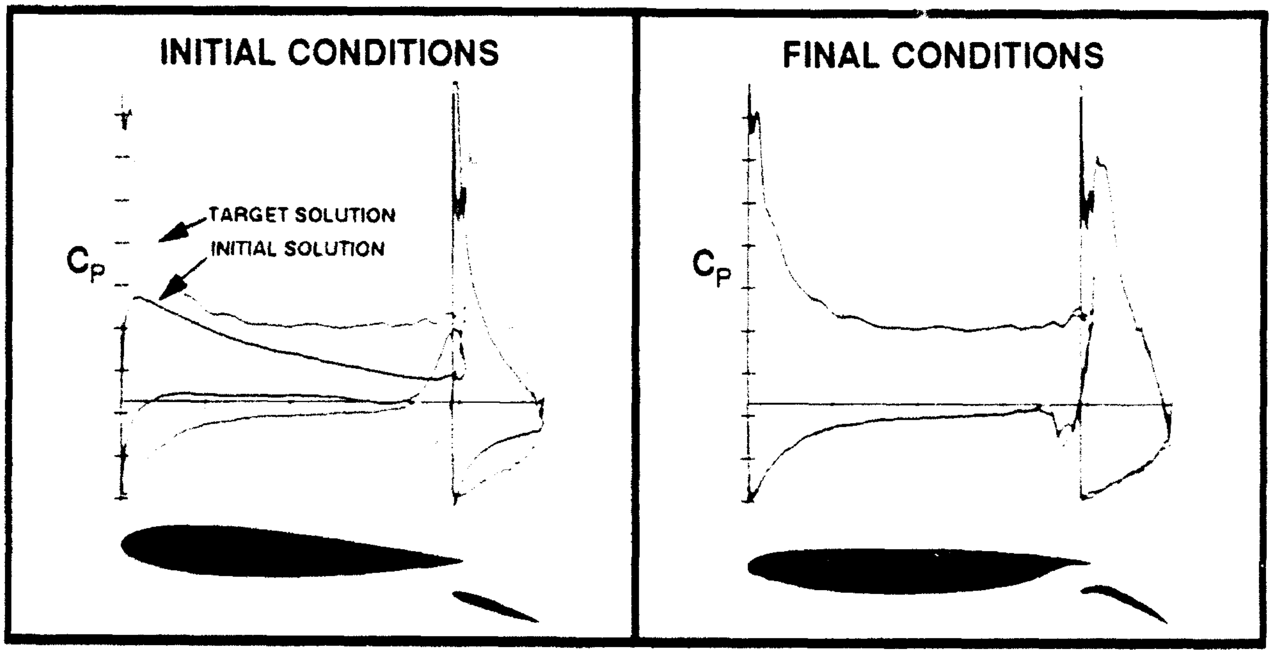
The program was developed and validated on several single-element airfoils. The first multiple-element airfoil test case was a Whitcomb slotted supercritical airfoil with a trailing-edge flap set at 20°. The target velocity distribution was obtained by direct Euler solution of the known Whitcomb geometry. The initial airfoil geometry was two symmetric (NACA 0014) airfoils with chord length and trailing-edge locations of the target geometries. The picture below shows the initial and final geometries and pressure distributions. A three-element airfoil test case which had a 30° leading-edge flap and a 30° trailing-edge flap was attempted, but proved to be too difficult a case for the current method.

Future Plans

A three-dimentional version of the inverse design procedure will be developed. The same MGM method and spring analogy will be used, however, a structured finite volume Euler solver based on Jameson's algorithm will be used for the flow solver. This program will require a two-block structured grid that is aligned streamwise over the wing. A simple wing-body test case will be used to develop and validate the code.

Publications

Kinsey, D.W., and Jolly, B.A.,"An Unstructured Approach to the Design of Multiple-Element Airfoils," AIAA Paper 92-2709, June 1992.



Two-Element Airfoil Inverse Design.

Three-Dimensional Unstructured Post-Processing

Lt Phillip M. Fite
Interdisciplinary and Applied CFD Section

Research Objective

Following grid-generation and flow-solution, the third critical stage to the CFD solution process involves post-processing, or the visualization of calculated data. Post-processing provides scientists and engineers with a graphics-based analysis tool for presenting CFD data as well as verifying the grid-generation and flow-solver applications. Due to their simpler generation and quicker adaptation characteristics, unstructured grids are becoming the choice in the CFD community as the need to analyze more complex geometries increases. Today, there are many post-processors for structured grids available, yet few meet the current demands for visualizing unstructured grid data.

Approach

An unstructured post-processing capability is being developed under contract to Vigyan, Incorporated (Dr. Paresh Parikh, Principal Investigator). Originally developed for NASA, Vigyan's code VPLOT3D was the only available post-processor capable of visualizing unstructured 3-D grids. Original versions of this code were limited to processing unstructured grid sizes up to 180,000 grid points. To allow the input of larger grids, Vigyan is modifying VPLOT3D by increasing the code's memory management capability and incorporating memory sharing techniques. To improve memory management, structured grids are handled separately from unstructured grids. Separating grid types reduces unnecessary data associated with structured meshes, and thus allows larger unstructured grid size. This increases available memory by reducing the need to calculate connectivity data except when needed. Incorporating memory sharing techniques such as computing derived quantities (Mach number, entropy, etc.) "on the fly" eliminates the need to carry those values in the processing arrays.

Accomplishments

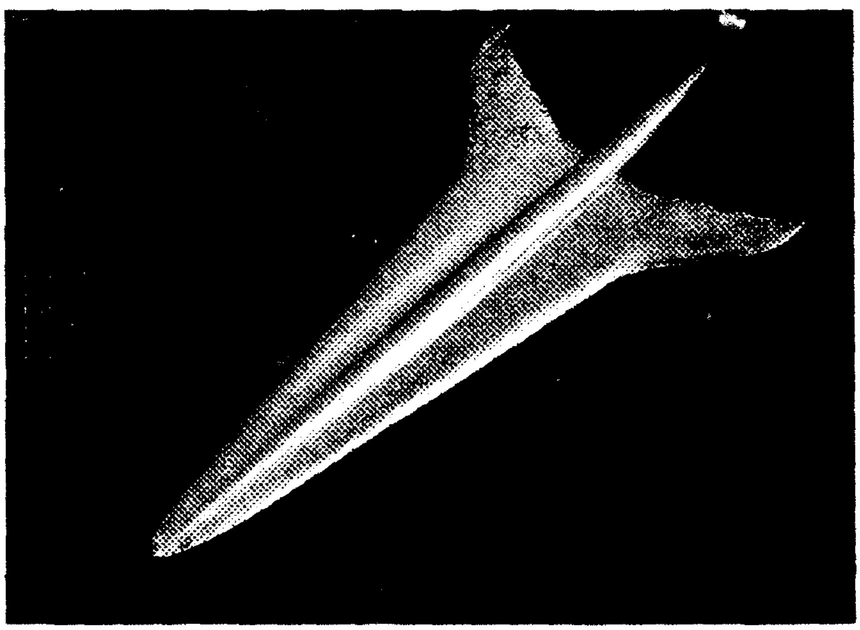
With the current options available, VPLOT3D's unstructured capability has improved memory allocation to process grid sizes of up to 250,000 grid points. In addition to grid size input, working versions of the code possess some unique features for unstructured processing. *These include arbitrary cutting planes, iso-volume* plotting, and input of multiple data formats.

Future Plans

To meet future Air Force needs, VPLOT3D will be modified to incorporate some additional features. These include portability to multiple platforms, time-dependent visual simulation (animation) of unsteady flows and moving objects, plotting time dependent data on a moving plane or grid, plotting experimental data, and store separation analysis.

Publications

Parikh, P., et al. "A Package For 3-D Unstructured Grid Generation, Finite Element Flow Solution and Flow Field Visualization," NASA Contractor Report 182090, September 1990.



High Speed Civil Transport with Cp contours in a 3-D Unstructured Grid

TOPDUUG - The Only Package for Design Using Unstructured Grids

Marvin C. Gridley
Airframe Propulsion Weapons Integration Section

Research Objective

The use of CFD in research and design can often be a time-consuming and somewhat esoteric "art" to scientists and engineers who may not be completely familiar with the inner workings of grid generators, flow solvers, and post-processors. Highly interactive computer programs which provide graphical feedback have proven to be great tools for use in research. This effort focused on the generation of a computer program designed to integrate several of the tasks in the CFD process into an interactive, graphical package which was easily used by any engineer or scientist, not just by CFD code developers.

Approach

Several different computer codes were being used to perform the tasks of grid generation and pre- and post-processing within the Aeromechanics Division. These codes were combined in TOPDUUG to create a general "cradle-to-grave" tool for use in 2D CFD analysis. Interactive menus and windows were added to increase functionality and utility. An interactive on-line help system was incorporated. Capabilities of TOPDUUG include:

1. Fully interactive boundary condition specification
2. Two-dimensional unstructured grid generation, including viscous grids
3. Flow solver job file generation
4. Flow solution visualization
5. Time dependent flow solution animation

Accomplishments

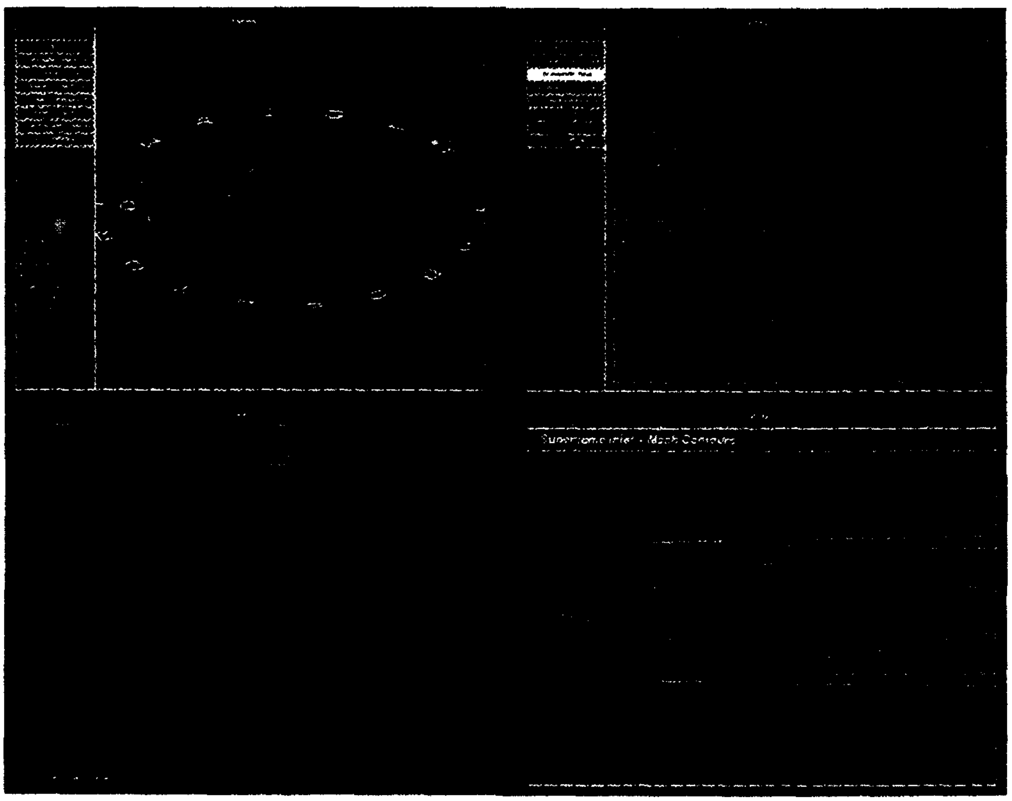
The first production version of TOPDUUG, The Only Package for Design Using Unstructured Grids, has been produced. TOPDUUG is a highly interactive 2D unstructured grid generator/pre/post-processor designed for ease of use and utility. TOPDUUG is currently being used in the Aeromechanics Division for several engineering applications as well as for code development.

Future Plans

Future plans include modification of the 2D unstructured grid generation module to be more robust and user friendly. Several scalar and vector functions will be added to the post-processing module to increase functionality. A user's manual describing operation of the software will be written.

Publications

Gridley, M. C., "TOPDUUG - The Only Package for Design Using Unstructured Grids: User's Manual," to be published.



Several Screens Showing TOPDUUG Functions

Numerical Simulation of Turbulent Cylinder Juncture Flowfields

Donald P. Rizzetta
CFD Research Section

Research Objective

This work investigates the ability of an advanced two-equation turbulence model to properly simulate flowfields which exhibit complex three-dimensional separations. Steady high Reynolds number subsonic and supersonic flows past a cylinder which is mounted upright on a flat plate are considered. These flowfields are characterized by formation of a horseshoe vortex system that wraps around the cylinder, and are of fundamental interest because of the intricate fluid phenomena which occur. In addition, the configuration is representative of a variety of practical situations including wing/fuselage and wing/pylon intersections, struts of aerospace vehicles, submarine conning towers, turbomachinery, wind tunnel model supports, architectural aerodynamics, and meteorological and geological applications.

Approach

Cylinder/plate juncture flowfields are simulated numerically by solution of the time-dependent three-dimensional compressible mass-averaged Navier-Stokes equations, which are integrated in time to achieve the steady state. Effects of fine scale turbulence are represented by a two-equation($k-\epsilon$) closure model which includes a generalized formulation and low-Reynolds number terms. For supersonic flow, the turbulence equations incorporate a compressibility correction.

Accomplishments

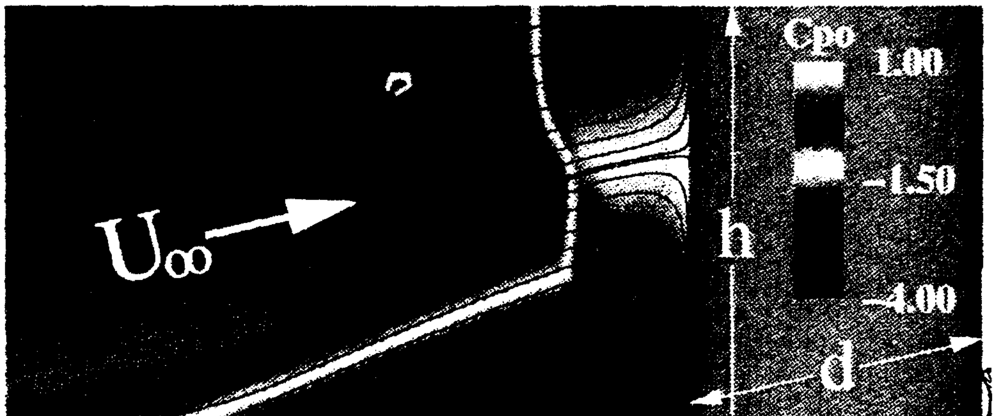
An existing vectorized three-dimensional implicit computer code was modified in order to accommodate solution of the turbulence equations, and the code was validated. Both subsonic and supersonic cylinder/plate juncture flowfields were generated numerically. All commonly observed physical features of the flows were reproduced by the calculations. A grid mesh step-size study was performed in order to assess resolution requirements of the solutions, and the effect of the compressibility correction was examined. Results of the computations have been compared to experimental data in terms of static and total pressure coefficients, velocity profiles, and surface limiting streamline patterns.

Future Plans

The turbulence model equations will be used to simulate the flow over a delta wing performing a roll maneuver, as well as the explosive rupture of an aircraft forebody.

Publications

Rizzetta, D. P. "Numerical Simulation of Turbulent Cylinder Juncture Flowfields," AIAA Paper 93-3038, July 1993.



Supersonic Cylinder/Plate Juncture Flowfield

Turbulence Modeling for Thrust Reverser Flow Prediction Methods

Kenneth E. Wurtzler
Interdisciplinary and Applied CFD Section

Research Objective

Thrust vectoring is one means to improving the maneuverability and take-off and landing performance of aircraft. However, thrust vectoring, especially in close proximity to the ground, can significantly increase the complexity of the flowfield around the aircraft. Problems that can result include hot gas ingestion, reduced lift, and foreign object damage to engines. To achieve good aircraft designs, it is necessary to have an accurate flow prediction method capable of resolving turbulence. This work addresses the issue of turbulence modeling for the inclined impinging jets associated with the use of thurst reversing in ground effect.

Approach

This effort combined numerical research with experimental results in order to produce a validated turbulence model. The cases studied were an impinging jet issuing from a nozzle normal to the ground plane and one issuing from a nozzle inclined 45° into the crossflow. Nielsen Engineering and Research, Inc. (Dr. Robert Childs, Principal Investigator) performed large eddy simulations (LES) to obtain mean flow and turbulence data. Basic physical experiments, performed at Stanford University, provided additional flow data.

This data was analyzed to identify the dominant physical processes which govern turbulence in these flows. Modifications to the k-e turbulence model were based on these findings. The new coefficients in the resulting modified turbulence model were determined via numerical optimization for benchmark jet flows.

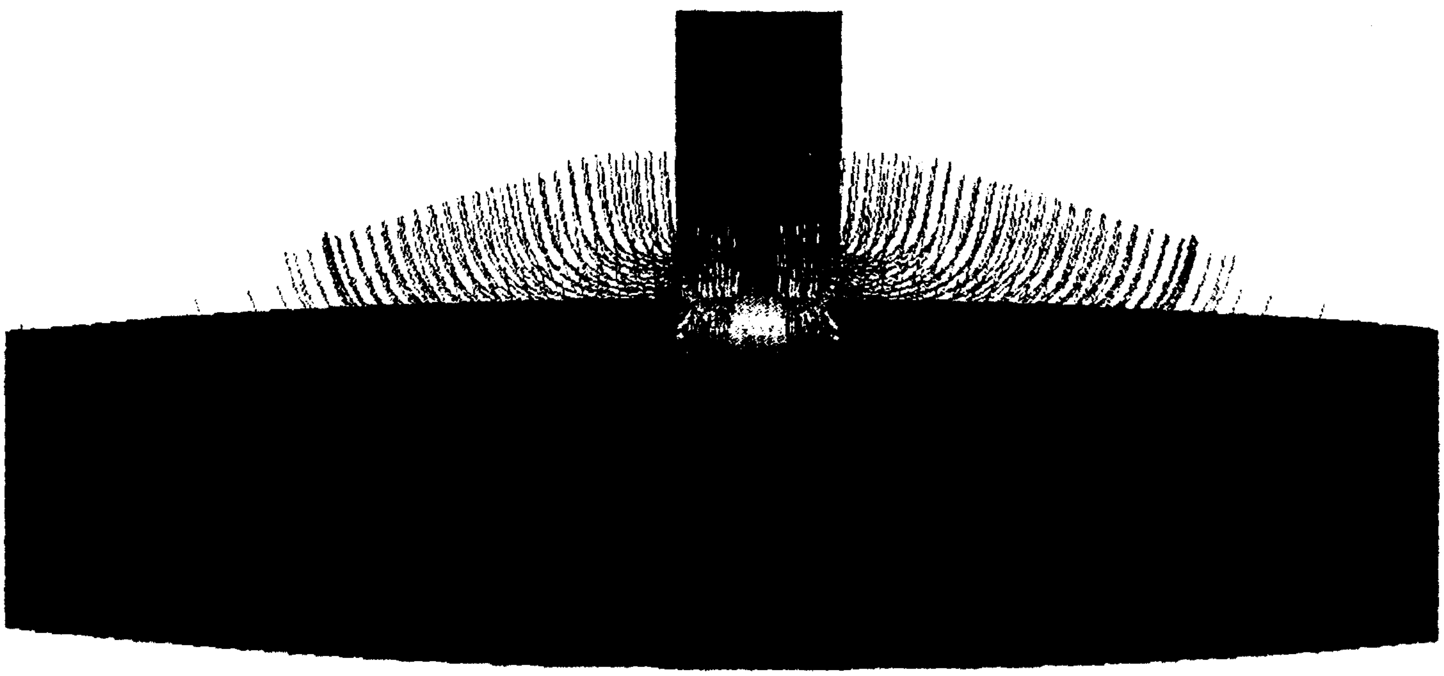
Accomplishments

Streamline curvature and vortex stretching were identified as the two most significant influences on turbulence. Modifications which account for these two features were added to the standard k-e turbulence model. In the cases considered, the modified model consistently improved the accuracy for the computed location of the ground vortex. Errors in the computed vortex location were as large as 50% in calculations with the standard turbulence model and could be of either sign. The modified model reduced these errors to the range of 13-18%.

Publications

WL-TR-92-3121, "Turbulence Modeling for Thrust Reverser Flow Field Prediction Methods", December 1992.

Childs, R. E. and Rodman, L. C., "Turbulence Modeling for Impinging Jet Flows," AIAA-92-2672.



Normal impinging jet, with Mach=1.0, exiting from a pipe into low-speed crossflow.

The Effect of Leading-Edge Cross-Sectional Geometry on Vortex Flow Aerodynamics

T. Sean Tavares - NRC Associate
CFD Research Section

Research Objective

The objective of this research is the detailed calculation of viscous flow features which govern vortex separation from the leading edges of highly swept wings at high Reynolds numbers. The extent to which leading-edge vortex separation occurs can have a strong effect on lift, drag, and pitching moment, and depends in part on the leading-edge cross-sectional geometry. For wings with sharp leading edges, the point of primary vortex separation is fixed at the leading edge. However, for wings having rounded leading edges, this is not necessarily the case, and the flow may remain attached along part or all of the leading edge depending on angle of attack, Reynolds number, and Mach number.

Approach

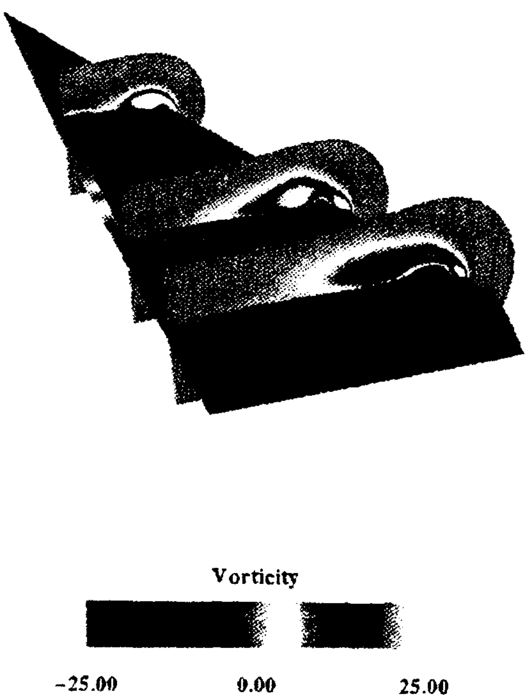
The approach taken is the computational solution of the 3-D Reynolds-Averaged Navier-Stokes equations. They are solved using the implicit, approximately factored Beam-Warming algorithm as implemented within an in-house computer code (FDL3DI). Turbulence closure is achieved with the Baldwin-Lomax algebraic turbulence model, which has been modified to account for the effects of large scale crossflow separation. A special challenge of this problem is to generate a grid with sufficient resolution to resolve the thin boundary layers characteristic of high Reynolds number flows and to maintain good grid quality in the rest of the domain, without using an excessive number of grid points.

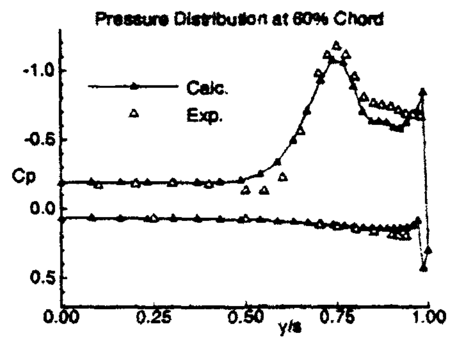
Accomplishments

The figure illustrates a calculation of the flow about a clipped delta wing with 65° leading-edge sweep and a sharp leading edge. The flow conditions are 10° angle of attack at Mach 0.4 and a Reynolds number of 9×10^6. Good agreement between the calculated and measured spanwise pressure distributions, as shown for the station at 60% of the root chord is indicative of the successful calculation of both the primary and secondary vortex structures. These calculations represent the highest Reynolds number turbulent flow cases run to date with the in-house Navier-Stokes code. In addition, comparisons of preliminary solutions obtained on coarse grids for both sharp and round-edged wings, reproduce observed dependences of vortex flow patterns on leading-edge geometry.

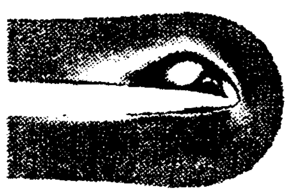
Future Plans

Calculations are underway on the 65° clipped delta with rounded leading edges, for which experimental measurements show separation along only part of the leading edge at 10° angle of attack. Calculations are planned for both geometries on refined computational meshes having 1.6 million grid points.





60% Chord



Calculated vortex flow structure above sharp-edged delta wing and pressure distribution comparison.

Vortex Breakdown Above a Pitching Delta Wing

Miguel R. Visbal
CFD Research Section

Research Objective

Vortex breakdown represents one of the unresolved issues in high-angle-of-attack aerodynamics. Further understanding of this complex fluid dynamics phenomenon is required for enhancing aircraft maneuverability and for alleviating undesirable fluid/structure interactions such as tail buffet. The purpose of this research is to study the transient behavior and 3-D flow structure of the leading-edge vortex breakdown above a delta wing which is pitched to a high angle of attack.

Approach

The flows are simulated by solving the unsteady three-dimensional full Navier-Stokes equations expressed in strong conservation law form. A general time-dependent coordinate transformation is incorporated to treat the prescribed wing motion. The discretized equations are solved employing the implicit approximate-factorization Beam-Warming algorithm. The accuracy and stability of the scheme are enhanced by means of a subiteration procedure.

Accomplishments

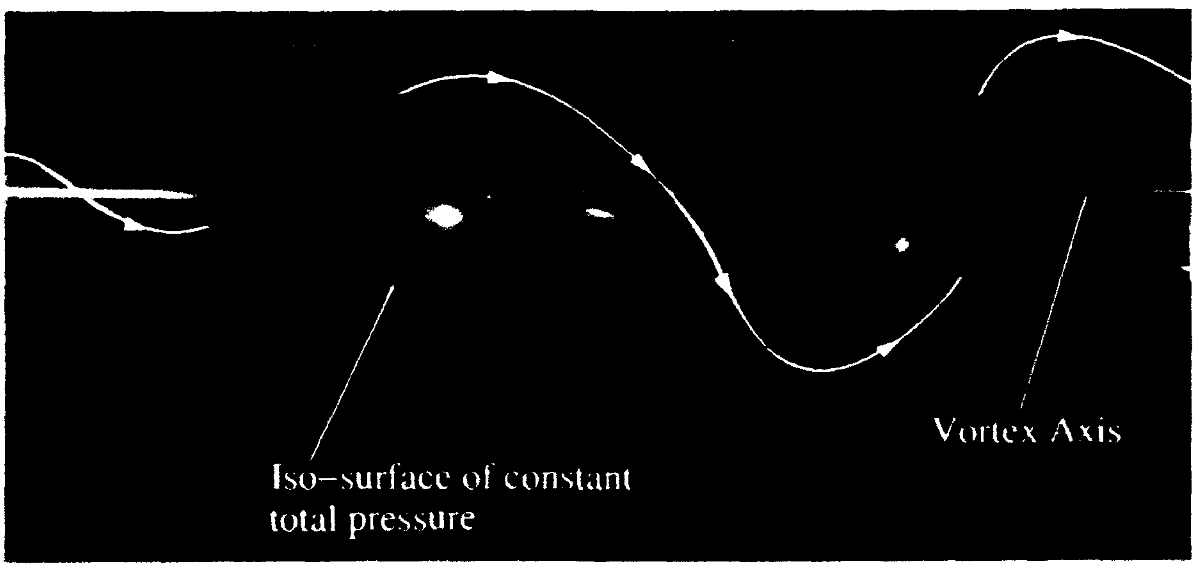
Calculations were performed for a 75° sweep delta wing pitching at a constant non-dimensional pitch rate $\Omega^+ = 0.3$, from an initial angle of attack $\alpha_i = 25°$ to a final angle $\alpha_f = 50°$. The pitch axis was located at the wing trailing edge. An assessment of the effects of numerical resolution and the favorable comparison with available experimental data indicate the computational approach captures the basic dynamics of the transient breakdown. The angular delay and onset of breakdown are found to be strongly linked to the pressure gradient prevailing along the vortex axis. A description of the 3-D instantaneous structure of vortex breakdown is provided for the first time using critical-point theory. The region of reversed flow in the vortex core is associated with the appearance of pairs of opposite 3-D spiral/saddle critical points. During its early stages, the vortex breakdown is fairly axisymmetric. However, as breakdown proceeds upstream, asymmetric effects become increasingly important and lead eventually to the formation of a breakdown bubble. This bubble structure is open and contains within itself a pair of stagnation points which are diametrically opposed and which rotate in the same sense as the base flow. A representation of the breakdown bubble using an iso-surface of constant total pressure shows a great deal of resemblance to experimental flow visualizations of axisymmetric vortex breakdown in a tube.

Future Plans

Further investigation of the relation between the different representations of the breakdown region (i.e. instantaneous flow structure, streakline visualizations and mean-flow measurements) will be pursued for unsteady breakdown above a stationary delta wing. Investigation of the parametric effects of pitch rate and pitch-axis location, as well as, compressibility effects is in progress.

Publications

Visbal, M. R., "Structure of Vortex Breakdown on a Pitching Delta Wing," AIAA Paper 93-0434, January 1993.



Computed bubble-type breakdown above pitching delta wing

Numerical Simulation of Delta-Wing Roll

Raymond E. Gordnier
CFD Research Section

Research Objective

The combat requirements for modern fighter aircraft necessitate that they be highly maneuverable and able to operate at high angles-of-attack. These requirements lead to an interest in understanding the aerodynamics of delta-wing maneuvers, including high-rate rolling motion. The focus of the present research effort is the numerical simulation of a constant roll-rate, "roll-and-hold" maneuver. The main objective is to describe the dynamical behavior of the vortex position and strength, as well as the corresponding effect on surface pressure, lift and roll moment.

Approach

The governing equations solved are the unsteady, three-dimensional, full Navier-Stokes equations in strong conservation form. A general time-dependent coordinate transformation is introduced to allow for the body motion. The equations are solved using the implicit, approximately factored Beam-Warming algorithm. Both a full block tridiagonal and diagonalized inversion schemes are available. A subiteration procedure is also incorporated to improve the accuracy and robustness of the algorithm.

Accomplishments

Two fixed-roll angle cases ($\phi = 0°$ and $\phi = 45°$) were computed to demonstrate the capabilities of the numerical scheme before computing the dynamic roll maneuver. The results showed excellent qualitative and quantitative agreement with experimental measurements. Then a constant roll rate, $\Phi = 0.1325$, "roll-and-hold" maneuver from $\phi = 0°$ to $\phi = 45°$ was computed. The dynamics of the vortices during the roll maneuver have been described. A lag in the upward leading-edge vortex similar to that observed for delta wing rock was noted. The effect of the dynamic motion on the vortex strengths has also been described. A simple, quasi-static explanation of this vortex behavior based on effective angle-of-attack and sideslip angle during roll has been proposed.

Future Plans

Current work on this project consists of the investigation of parametric effects on the aerodynamics of the roll maneuver. The effects of roll rate, Reynolds number, and asymmetric initial conditions (i.e., starting from nonzero roll angles) are being studied. Calculations of turbulent flows over delta wings using a two equation k-ϵ turbulence model are also planned. The ultimate objective of this work is to be able to numerically simulate delta-wing rock both with and without vortex breakdown.

Publications

Gordnier, R. E. and Visbal, M. R., "Numerical simulation of Delta-Wing Roll," AIAA-93-0554, January, 1993.

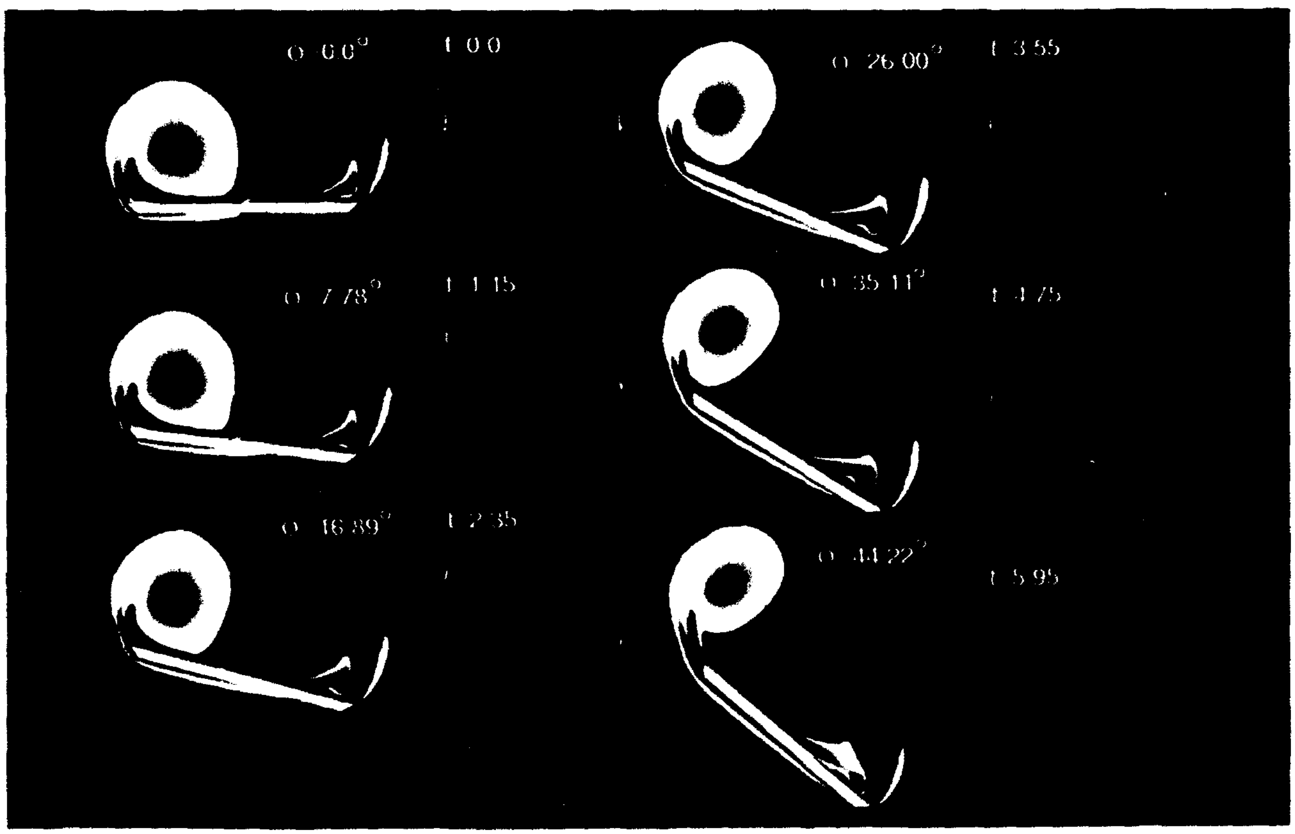


Figure 12. Vortex Dynamics during a Roll Maneuver: $X/L = 0.9$

Nonequilibrium Hypersonic Flowfield Prediction for Flow Past Blunt Bodies

Eswar Josyula
CFD Research Section

Research Objective

To perform numerical modelling of the flowfield using the Navier-Stokes equations with finite rate chemical kinetics applicable to high-temperature air for hypersonic flows in chemical and thermal nonequilibrium.

Approach

The axisymmetric Navier-Stokes Equations, 5 species continuity equations, 3 vibrational energy equations, and a total energy equation are solved in a fully coupled explicit method using the Roe Flux-Difference Split Scheme.

Accomplishments

The extension of the Roe scheme to include the finite rate chemical kinetic equations follows the approach of Grossman and Cinnella. Second order accuracy is achieved with the MUSCL approach. The full Navier-Stokes equations are solved with finite rate chemistry for the flow past an axi-symmetric blunt body at zero incidence at several Mach numbers ranging from Mach 10 to 18, at altitudes of 22 km and 37 km. The wall temperature is set at 555 K and the maximum shock temperature is around 15,000 K. Both non-catalytic and fully catalytic wall boundary conditions for the species are utilized.

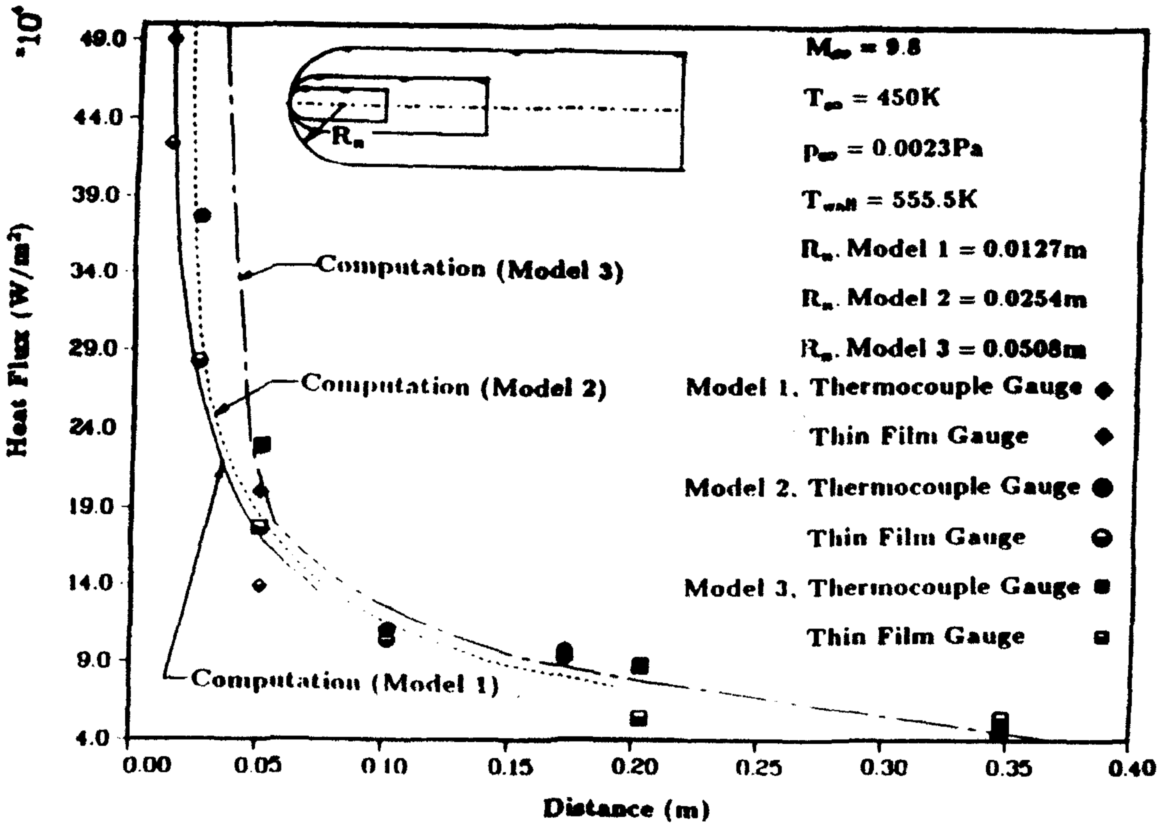
The air simulated in the modelling is composed of atomic oxygen and nitrogen, molecular oxygen and nitrogen and nitric oxide. The chemical source terms are derived from the Law of Mass Action. The importance of a multi-temperature model for the proper treatment of the vibrational relaxation in the prediction of the surface heat transfer is shown. The variation in computed shock-standoff distance substantiates that the Mach number independence principle applicable to ideal gases does not apply to dissociating flows. The computed stagnation point heat flux and the surface heat transfer show excellent comparison with the experimental data obtained from recent experiments conducted in a shock tunnel and from a set of earlier shock tube experiments.

Future Plans

Effects of ionization and radiation become important at higher temperatures and they will be studied for proper prediction of the flowfield and surface heat transfer on the flow past blunt bodies.

Publications

Josyula, Eswar and Shang, Joseph "Computation of Nonequilibrium Hypersonic Flowfields Past Hemisphere-Cylinders." Journal of Thermophysics and Heat Transfer, to be publisheu in July 1993.



Comparison of surf: ce heat transfer distribution with shock-tunnel experiments for three different-sized models

Stability Analysis of a Combined Couette-Poiseuille, Two-Fluid Flow

Lt John J. Nelson
Interdisciplinary and Applied CFD Section

Research Objective

In this work, the complete theory is developed for the linear stability analysis, energy analysis and weakly nonlinear analysis for a combined Couette-Poiseuille, two fluid flow. The linear stability analysis determines the conditions at which the basic, laminar flow will become unstable and transition occurs. The energy analysis determines the mechanisms which cause the instability. In multi-fluid incompressible flow, instability is influenced by five mechanisms: viscous dissipation, Reynolds stress, interfacial friction, density stratification and surface tension. Weakly nonlinear analysis determines if standing wave solutions are possible in the flow ensuing after transistion, and if so, determines the transient amplitude of the waves.

Approach

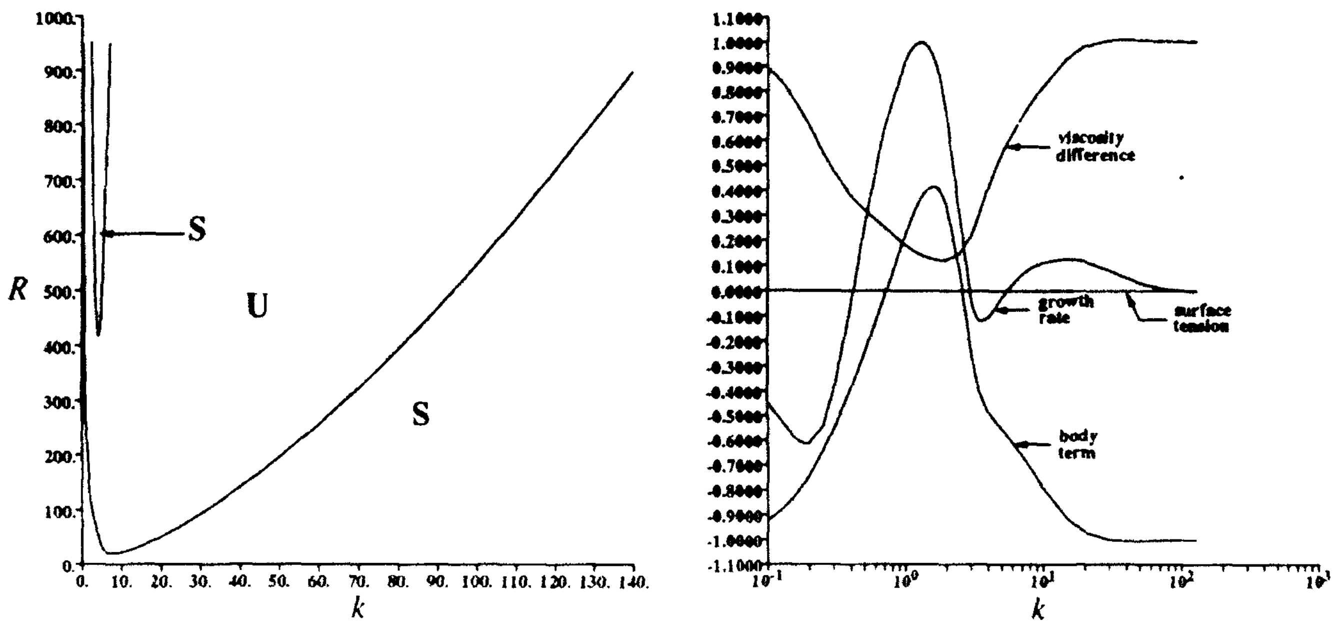
The linear stability analysis is accomplished by linearly perturbing the continuity, Navier-Stokes and interfacial equations. A separable solution is assumed, which leads to a complex eigensystem problem, which determines the growth rate of infinitesimal disturbances in the flow. Full viscous and surface tension effects are included. The results of the linear analysis are used to derive a energy analysis. The weakly nonlinear analysis is accomplished by expanding in a small parameter to third order about the basic flow. Systems governing the distortion of the mean flow, the rise of the second harmonic and growth rate of finite amplitude waves are found and solved.

Accomplishments

This work completely derives a theory and provides test case solutions for transition prediction coding of incompressible flows. The flow situations studied in the work are the standard test cases for linear stability code accuracy. Our linear stability analysis is in agreement with the standard answers. Going beyond linear analysis, this work is the first to identify the mechanisms leading to the instability. Specific questions relating to normalizations are also addressed for the first time.

Future Plans

The theory developed is being employed in an incompressible Navier-Stokes code and used to study a problem of current interest. Also planned is the development of a stability theory for compressible flows. This theory will be used to give transition prediction capability to our in-house compressible Navier-Stokes code.



(a) (b)

(a) Linear stability analysis of two fluids in a Couette flow. R represents the Reynolds number and k the wave number. (b) Energy analysis for the flow of (a) at $R = 600$. The term "body term" represents viscous dissipation plus Reynolds stress.

Computational Aerodynamic Analysis of a Decoy Configuration

Frank C. Witzeman Jr
Interdisciplinary and Applied CFD Section

Research Objective

Aerodynamic analyses of air vehicles operating over a range of flight conditions can be performed by a variety of methods including analytical, semi-empirical, Computational Fluid Dynamics (CFD), and experimental testing. Limitations in both the semi-empirical and testing techniques for a decoy configuration lead to the use of CFD to fill gaps in the required data. Specifically, aerodynamic data at transonic, high angle-of-attack conditions were needed. Primary interest was in obtaining in-plane (pitch) forces and moments due to inertial effects (inviscid flow). Effects due to roll angle and small changes in Mach number and angle-of-attack were of interest, but secondary.

Approach

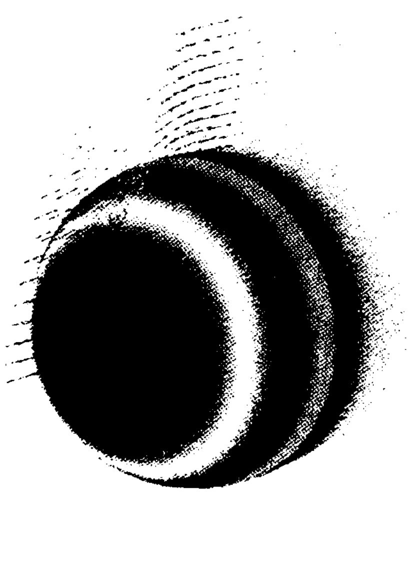
The in-house Euler code MERCURY was used to obtain CFD solutions for the decoy. A structured grid consisting of 45 blocks and approximately 720,000 grid points was generated from the GRIDGEN code. The blocking scheme was beneficial in reducing computer memory requirements where the maximum memory required depends only on the largest block. Integrated and component forces and moments were compiled for each solution. The components consisted of the nose, centerbody, aft-body, and tail. The integrated and component results were compared with the semi-empirical method results and experimental data (integrated only). The final solution matrix consisted of one roll angle, three Mach numbers, and several angles-of-attack.

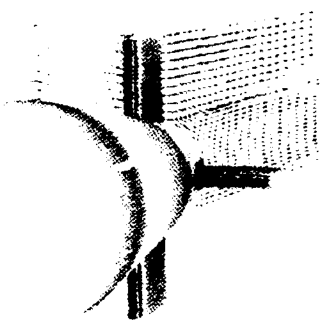
Accomplishments

Euler solutions for the decoy at Mach numbers of 0.5 and 0.7 have been obtained for several angles-of-attack and 0° roll angle. The low-speed results compare well to the experimental data up to intermediate angles-of-attack (20°), whereas the Mach-0.7 results show discrepancies beyond 10° angle-of-attack. Compressibility effects have a major impact on the aerodynamics as noted in the CFD results. These results also indicated that the semi-empirical method did not accurately predict center-of-pressure location, especially for the aft-body component.

Future Plans

Additional CFD solutions at Mach-1.2 and angles-of-attack up to 90° are planned. These results are lacking in the experiments. All results will be tabulated for the total integrated and component forces and moments. Compressibility effects at transonic conditions will be investigated in more detail with the CFD solutions.





Mach Number Contours and Velocity Vectors for Decoy Configuration

Euler Analysis for Delta Wing Design

Lt John S. Seo
Interdisciplinary and Applied CFD Section

Research Objective

At the request of ASC/XR, the feasibility of a 14% thick flying delta wing to sustain efficient flight at a design speed of Mach 0.90 and lift coefficient of 0.60 was tested. CFD analysis was applied for this configuration as the calculation of wave drag using empirical methods were unreliable. This delta wing has a leading-edge sweep angle of 53.1 degrees and half-wing planform area of 525.87 square feet.

Approach

Since the parameter of interest was wave drag, an Euler solution was sufficient for this analysis. Three cross-sectional cuts of the wing were prepared in an interactive graphics grid generator, I3G/VIRGO, to create a full three-dimensional geometric surface definition. A single C-H type grid with dimension of 61x40x70 was created using another interative multiple-block grid generator, GRIDGEN. MERCURY, an in-house 3-D Euler flow solver based on Jameson's algorithm, was used for wave drag calculations.

Accomplishments

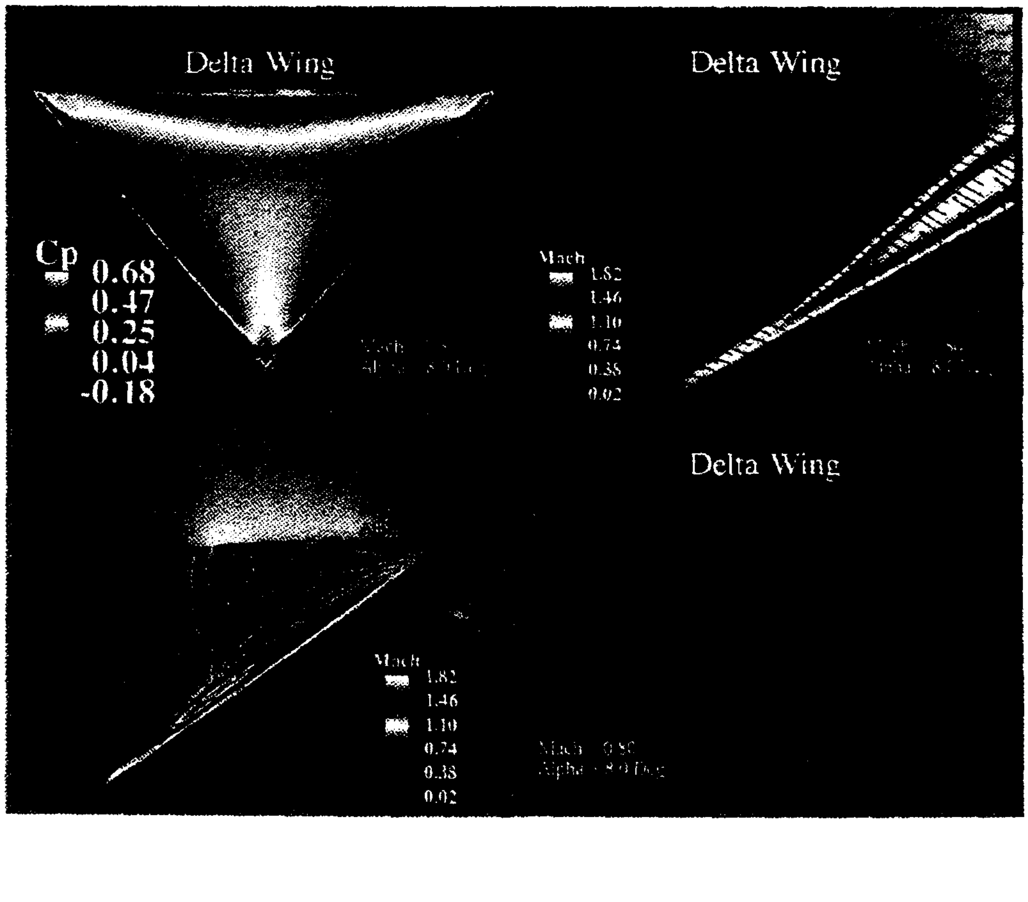
Over 50 solutions with Mach number ranging from 0.5 to 0.95 and angle-of-attack ranging from 0 to 15 degrees were computed on two of AFMC's supercomputers: Wright-Patterson AFB's Cray/XMP ,and Eglin AFB's Cray/YMP. It was found that at Mach = 0.90, and even at a lower Mach of 0.80, this 14% thick delta wing will not achieve the desired C_l of 0.60. A strong shock slightly past the leading edge, and reversed flow near the wing tip contribute to the C_l loss.

Significance

Existing CFD tools allow WL/FIMC to perform an Euler study for this relatively simple configuration with ease. This particular study lasted 5 weeks from beginning to end.

Presentations

The results of this analysis were presented at the Dayton-Cincinnati AIAA mini-symposium, 25 Mar 93.



MERCURY solution for delta wing at Mach 0.80 and 8 degrees angle of attack

Numerical Simulation of a Jet Expulsion from an Aircraft Forebody, With Comparison to Experiment

Capt James Mundy - CFD Research Section

Research Objective

This research investigates the effect a sonic, underexpanded jet issuing from a generic aircraft forebody, has on the airflow around the aircraft, and on the resulting aerodynamic forces.

Approach

An implicit Beam-Warming finite difference scheme is being used to solve the Navier-Stokes equations around the aircraft forebody. A $k-\epsilon$ turbulence model is used to account for the effects of turbulence. The resulting numerical solutions are compared with pressure data and flow visualization video and still photography taken from wind-tunnel tests. Pressure data is in two forms, the first being pressure tap data, the second being data taken from a new pressure sensitive paint technique.

Accomplishments

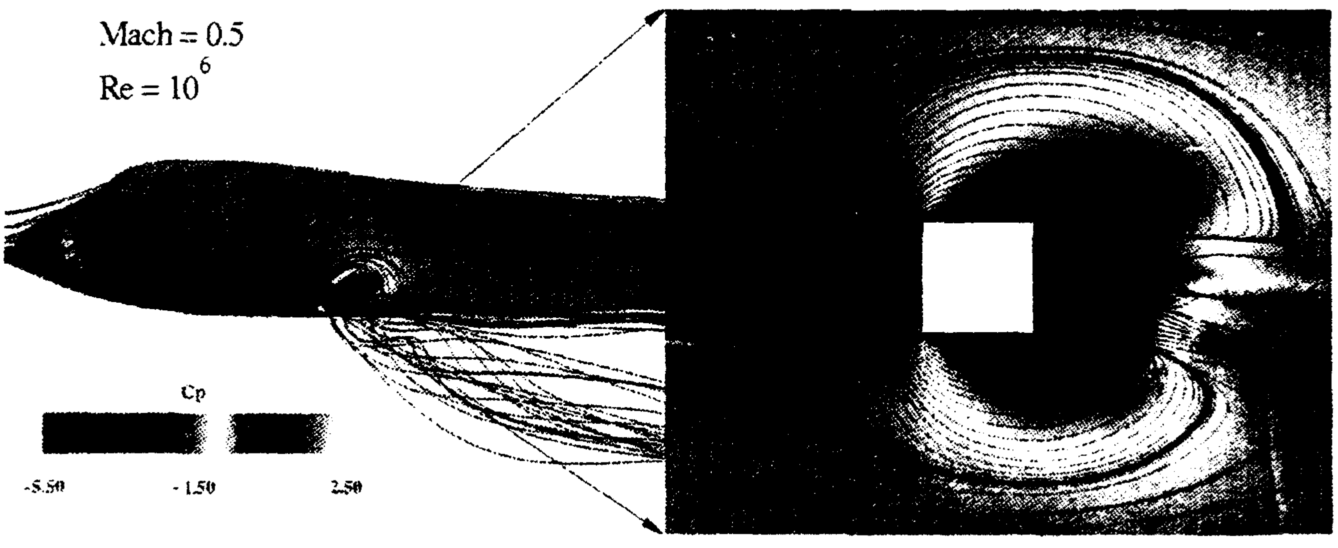
The three-dimensional Navier-Stokes equations, with a $k-\epsilon$ turbulence model have been solved around the generic aircraft forebody on a 108x88x43 grid (408,672 grid points), without the jet blowing. Agreement between the numerical solution and experimental data is excellent. The solution for the jet-on case is still only partially converged, though indications are that the numerical solution will agree closely with the experimental data.

Future Plans

The jet-off case will be refined using a finer grid (approximately 1.25 million grid points) to provide validation for the CFD scheme. Once this is done, the generic aircraft forebody will be modeled to conform to typical flight characteristics of a jumbo-jet. The free-flight case will model flight at Mach 0.75, at an altitude of 30,000 feet.

Publications

Melville, Reid. "Numerical Simulation of a Sonic, Underexpanded Jet from an Aircraft Forebody" FAA Aircraft Hardening and Survivability Symposium, August 1992, Atlantic City NJ.



Streamlines Around a Jet Issuing from a Fuselage

Numerical Analysis of a Chined Forebody with Asymmetric Strakes

Kenneth E. Wurtzler
Interdisciplinary and Applied CFD Section

Research Objective

Vortex forebody control (VFC) is increasingly being viewed as a way to provide greater directional control of fighter aircraft at high angle of attack. The major objective of this work to examine the effectiveness of various strake sizes on the yawing and rolling moments on the Fighter Lift and Control (FLAC) forebody alone. A secondary purpose is to validate the use of an Euler code for the prediction of flow over a chined forebody at high angles of attack.

Approach

The geometry utilized was the full FLAC forebody with an attached sting extending behind the final fuselage cut. Different sized strakes were attached to the left side at the apex of the nose. A 1.7 million point, multi-block grid was constructed using I3GVIRGO. In order to simplify the grid generation and to provide for quick substitution of different strake sizes, the ability to create various strake sizes was built into the grid. Grid spacing normal and tangential to the surface was kept equal so as to produce a dense, equally spaced gridded domain capable of capturing the core of the vortex. The flow past the forebody was computed by using an in-house Euler code, MERCURY. The full forebody was modeled so as to capture any asymmetry. Convergence on the forebody alone case was reached in about 2,000 iterations. Each strake case needed approximately 3,000 more iterations. Convergence was determined by a drop of four orders of magnitude of the residuals and leveling off of the body forces.

Accomplishments

Solutions were obtained at 15, 25, 35, and 45° angle of attack at a Mach number of 0.18. Initial calculations on the clean forebody resulted in no side force being produced, which is theoretically correct for Euler codes. The results showed that a short, narrow strake produced a yawing force that acts in the direction of the strake side. Wider strakes pushed the vortex away from the forebody, creating a yawing force that acts in the direction of the other side. The strong vortex that separates from the chined forebody is captured well by the Euler code.

Future Plans

Numerical analysis of normal and tangential forebody blowing will be simulated and compared to the asymmetric strake results. Additionally, an unstructured grid approach will be used to compare Euler and Navier-Stokes results for strake and blowing applications.

Publications

Wurtzler, K. E. "Numerical Analysis of a Chined Forebody with Asymmetric Strakes," AIAA Paper 93-0051, January 1993.

Vorticity contours and particles traces over FLAC forebody with largest and smallest strake at 45° angle of attack.

Inviscid Hypersonic Flow Around a Conically-Derived Waverider

Capt Gregory O. Stecklein
Interdisciplinary and Applied CFD Section

Research Objective

The current high level of interest in hypersonic flight, brought about to a large extent by programs such as the National Aerospace Plane, has focused much attention on the stringent requirements of aerodynamic design at these demanding flight conditions. The major objective of this research is to analyze the inviscidly dominated flowfield of a waverider forebody for on- and off-design flight regimes.

Approach

The governing equations are the three-dimensional Euler equations utilizing a finite-volume solver using spatial Roe-averaged, flux-difference splitting. They are solved with a standard McCormack explicit time integration scheme. Convergence is accelerated at the expense of time accuracy by the local time-stepping technique.

Accomplishments

Three general classes of solutions were investigated for two levels of grid refinement. Inviscid solutions were obtained for on-design conditions of Mach 10, 30.5 km pressure altitude and zero degrees angle-of-attack (AoA) and two off-design flight regimes. Qualitative results illustrated excellent shock capturing and flowfield formation characteristics with grid refinement.

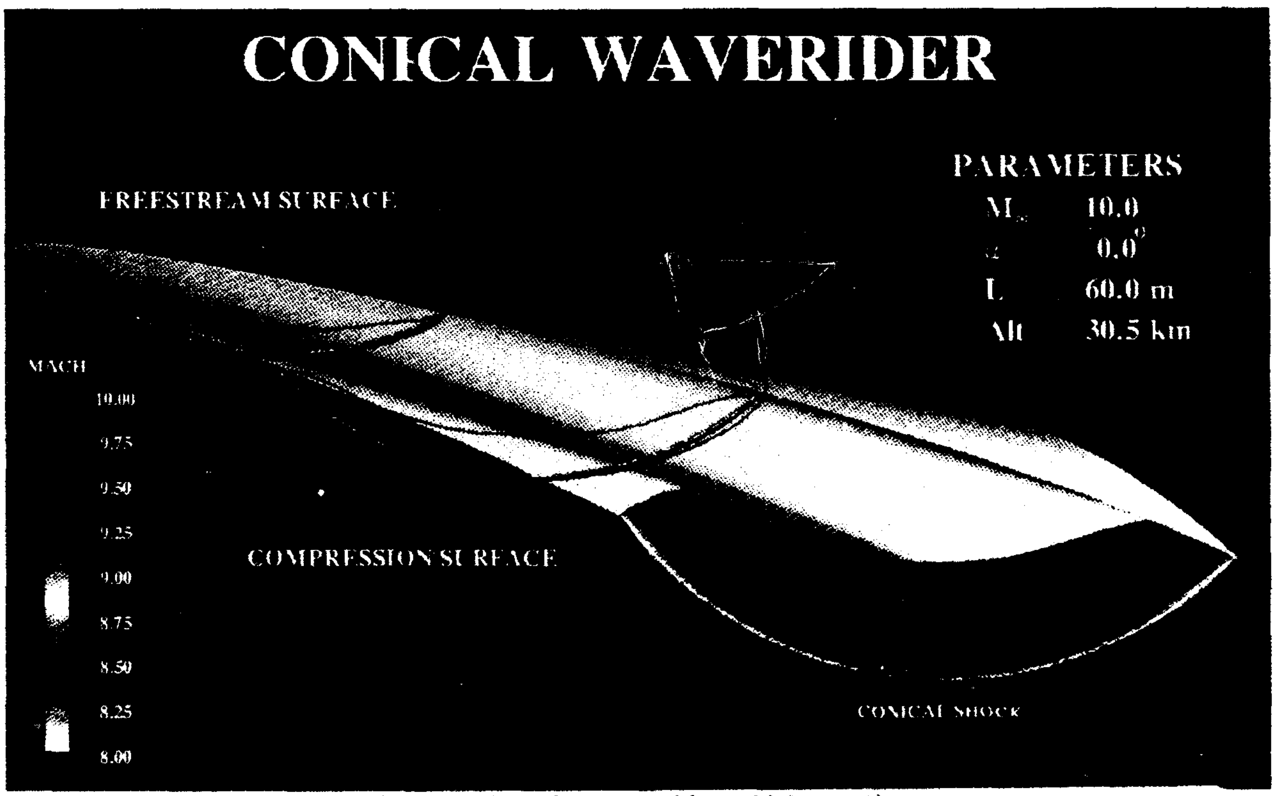
The on-design solutions were then validated against the analytical results which were based on hypersonic small disturbance theory, HSDT, and an integral momentum development. Quantitative results matched well with the analytic results to within 5% for lift and drag coefficients. Off-design calculations were run for the waverider at Mach 10 for an AoA range of ±5°, and at 0° AoA and a Mach range of 6 to 15. For the inviscid calculations, the off-design perturbations were characterized by wave drag effects. Of particular interest was the linear wave drag bucket associated with perturbations to the on-design attitude. For the range of attitudes investigated in this research, wave drag increased approximately 10% per degree angle-of-attack.

Future Plans

Current plans are to reevaluate this waverider configuration with a mass-addition leading edge. The implicit, full Navier-Stokes mode of the current algorithm will also be employed to investigate the effects of shock wave and boundary layer interaction.

Publications

Stecklein, G. O., "Numerical Solution of Inviscid Hypersonic Flow Around a Conically-Derived Waverider," AIAA Paper 93-0320, January 1993.



Inviscid Mach Contours: $M_\infty = 10.0; \alpha = 0°$

The Flowfield Past the X24C Reentry Vehicle

Datta Gaitonde - Universal Energy Systems. Inc
CFD Research Section

Research Objective

The purpose of this research is to examine the flowfield about complete lifting-body configurations at hypersonic speeds utilizing an accurate and efficient numerical algorithm. The ability of a high resolution scheme to provide accurate engineering estimates of surface thermal and mechanical loading is examined.

Approach

The full 3-D compressible Navier-Stokes equations in mass-averaged variables are solved in a finite volume formulation. Roe's flux-difference split method is used for the inviscid fluxes while viscous fluxes are centered. The implicit numerical algorithm is based on Gauss-Seidel line relaxation. Turbulence closure is achieved with the Baldwin-Lomax algebraic eddy viscosity model.

Accomplishments

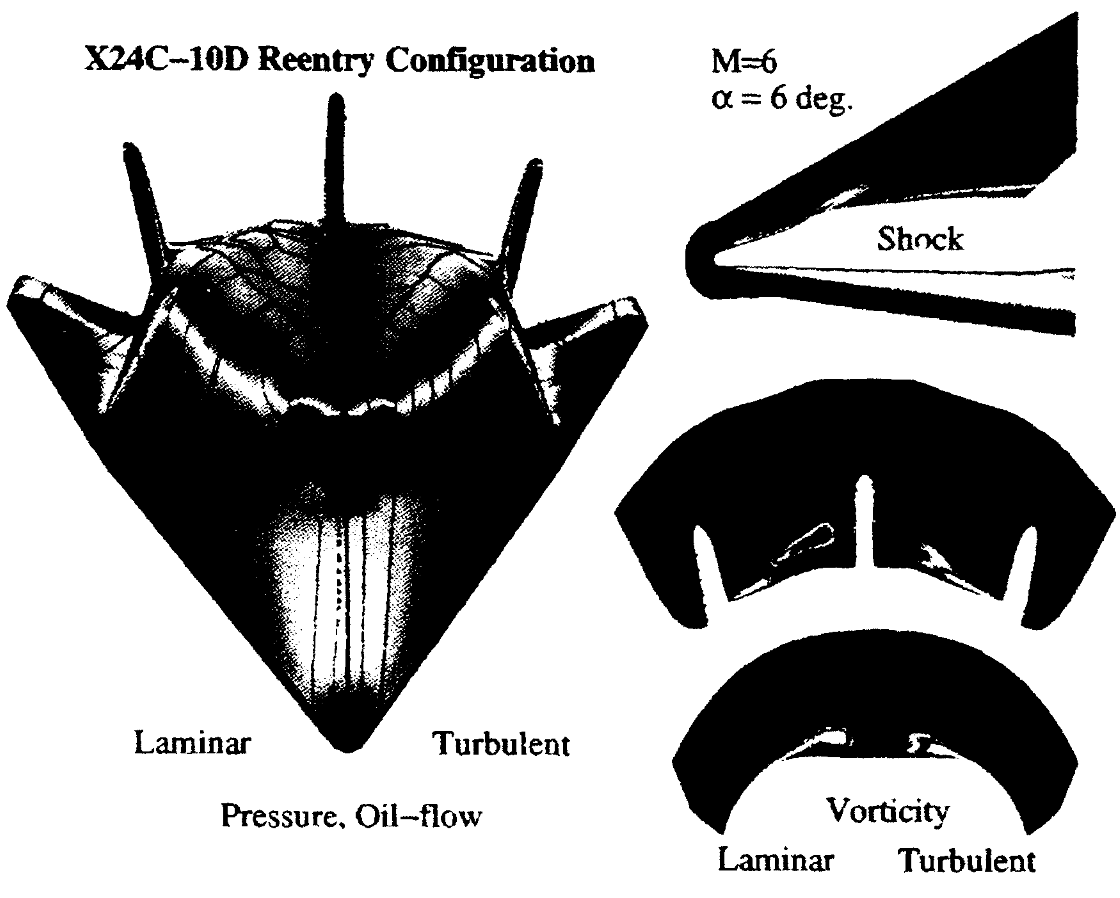
The choice of algorithm is determined with a critical examination of MUSCL based higher order upwind methods and a range of limiters. Specific emphasis is placed on the prediction of heat transfer rates in viscous flows. The flow past the X24C-10D configuration is then solved with a mesh of roughly half million grid points. The so-called "carbuncle" anomaly in the nose region is eliminated by a combination of different splittings without affecting the overall accuracy. Analysis of the computed turbulent flowfield show good agreement with experimental surface pressure and heat transfer data. Several aspects of the computed laminar and turbulent flowfields are investigated, including the interaction of the vortex structure with the tail fins.

Future Plans

The theoretical model will be extended to incorporate high temperature effects such as thermo-chemical non-equilibrium and ionization. A 2-D version of the code, employing non-equilibrium five-species chemistry and a simple harmonic oscillator model for vibrational energy excitation, has already been developed.

Publications

"Numerical Simulation of Flow Past the X24C Reentry Vehicle." D. Gaitonde and J. Shang. AIAA 93-0319. 31st Aerospace Sciences Meeting & Exhibit. Reno 1992.



Computed flowfield about X24C reentry vehicle

Calculations over a Swept Wing with a Gap in High Enthalpy Air

Denis P. Mrozinski and Capt Tom McClure
Interdisciplinary and Applied CFD Section

Research Objective

To investigate the affect of a leading edge normal surface gap in the skin of a swept wing on the flowfield and surface heat transfer at wind-tunnel conditions.

Approach

The HYLDA code (Hypersonic Low Density Analysis) was used in this investigation. HYLDA solves the full 3D Navier-Stokes equations set fully coupled with chemistry source terms. Preliminary calculations defined the general character of the flowfield, showed that the influence of a spherical nose cap was negligible, and helped to define the grid density requirements. Using this information, computational grids for a leading edge and wing section with a gap were constructed so that the flowfield around the gap was not influenced by the spherical nose cap or boundary conditions, and so that computation time was minimized.

Accomplishments

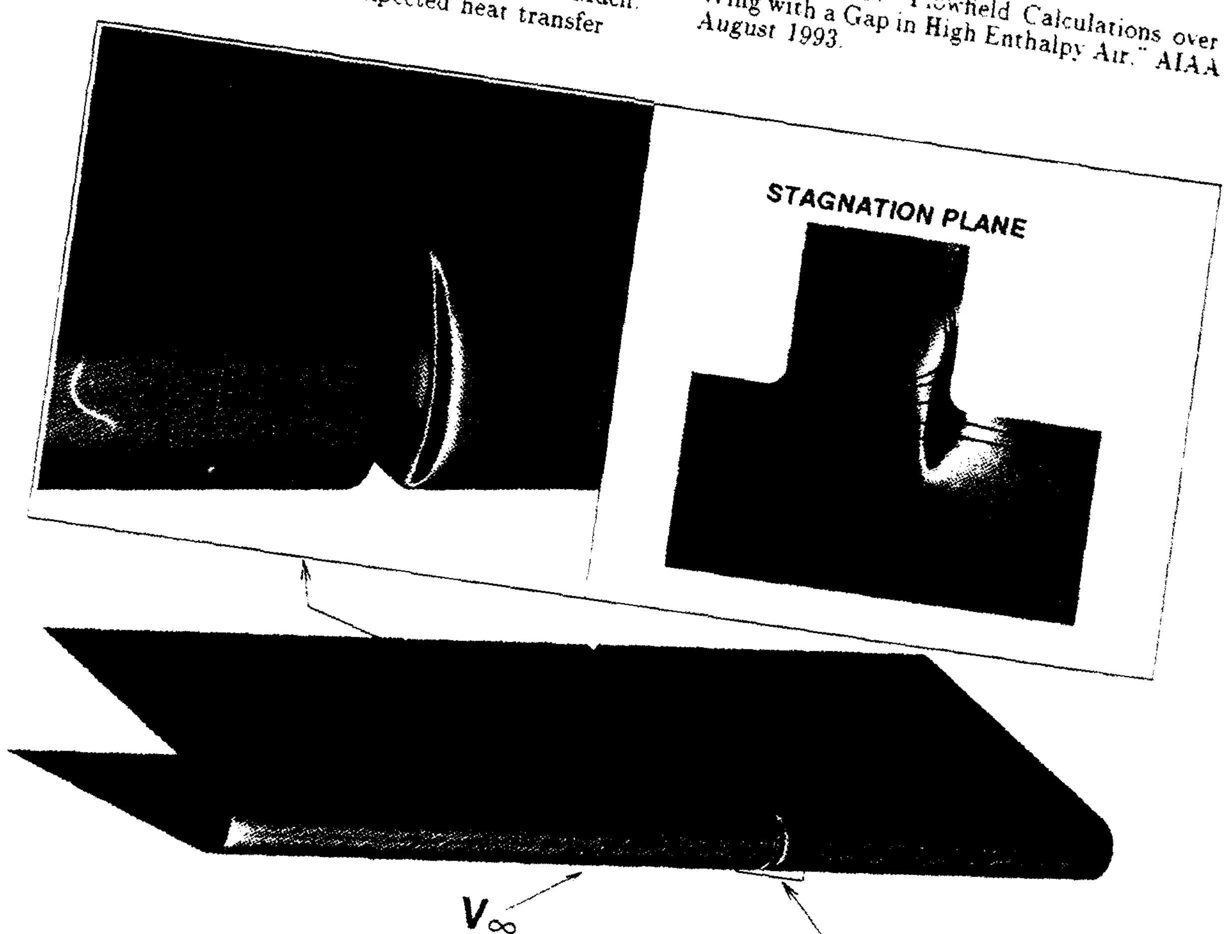
The flowfield about a swept wing section was calculated at wind- tunnel conditions (high enthalpy, low Mach, equilibrium air). Higher than expected heat transfer was observed on the upstream wallof the gap. A close look at the flowfield in the stagnation plane shows that a vortex is present in the gap and that the flow separates as it turns the upstream corner. The high heating is a result of the separated region near the upstream corner and the vortex impinging on the gap wall. The heating due to vortex impingement dissipates as the vortex moves out of the gap near the apex of the leading edge.

Future Plans

This study represents the only work relating to detailed heat transfer and flowfield analysis of surface gaps on a wing section for the NASP program. The current data will be compared with data taken from a planned wind tunnel test. Further analysis will be accomplished at an appropriate flight condition to determine if similar heating rates are encountered.

References

Mrozinski, D.P. "Flowfield Calculations over a Swept Wing with a Gap in High Enthalpy Air," AIAA 93-3489, August 1993.



Pressure contours on a 45 degree swept wing section with a gap

Bomb Blast Simulation

William Z. Strang
Interdisciplinary and Applied CFD Section

Research Objective

Simulate air blast loads induced by the detonation of a small charge.

Approach

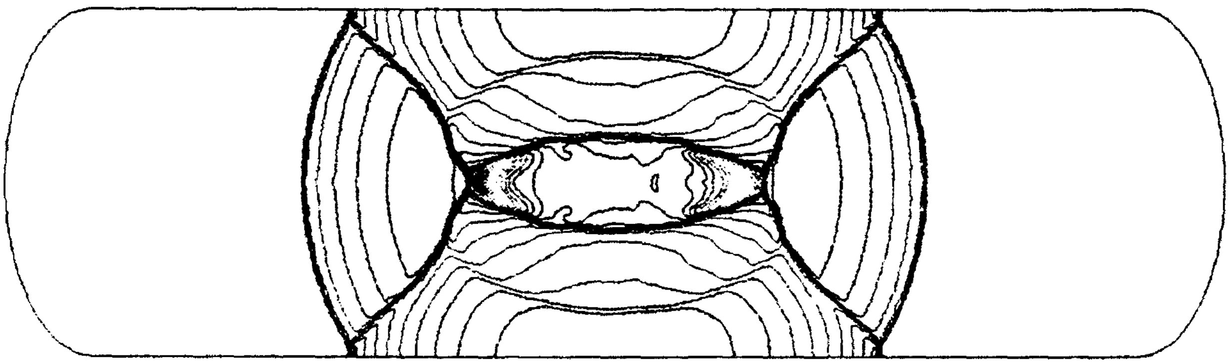
The general approach taken was to incorporate a bomb model into an existing computational fluid dynamics (CFD) code. COBALT was chosen as the CFD code to host the bomb model. This code, an unstructured grid, higher-order Riemann solver, was chosen because of its robustness and the ease with which the bomb and its surroundings could be discretized. The bomb model is very simple owing to the assumption that the detailed physics of charge detonation are unimportant at distances large compared to the charge radius. A corollary to this assumption is that the mass of the detonation by-products is small with respect to that of the surrounding medium. With these assumptions, a charge may be simply modelled as a mass and energy source. The mass and energy transfer rate is assumed constant and is calculated from tabulated detonation velocities for the charge material at hand. Owing to the high inertial forces involved, viscous effects were not simulated. Finally, radiation effects were assumed negligible due to the short time scale involved.

Accomplishments

One of the Aircraft Loading Tests conducted by WL/FIV was simulated. In the test, a bare spherical charge was located at the centroid of a rigid, right-circular cylinder with semi-oblate spheroidal endcaps. The particular test simulated was chosen for its accurate pressure histories at several recording locations. The simulation covered the first 11 milliseconds of the blast event. The accompanying figure depicts the pressure contours within the cylinder 1.56 milliseconds after detonation. In it, the primary blast wave, first sidewall reflection and implosion shock are seen. Comparisons between experimental and numerical pressure histories at various recording locations show excellent agreement.

Future Plans

We expect to simulate a detonation within a rigid, three-dimensional fuselage.



Pressure Contours at 1.56 milliseconds after Charge Detonation

Parallel Processing for Computational Fluid Dynamics

Stephen Scherr
CFD Research Section

Research Objective

This effort describes the parallel structure of an explicit Navier-Stokes algorithm and its implementation on a massively parallel message-passing computer.

Approach

The analysis of complex fluid flows and the future coupling of CFD techniques in interdisciplinary applications require ever increasing amounts of computational power. Trends in computer architecture indicate that the required computational power will only be available through the use of many processors all working together to perform the analysis. The architectures which currently show the most promise are those in which each processor has its own private memory and communicates with other processors by passing information through messages. Samples of these are tightly coupled massively parallel computers, and loosely coupled networks of workstations.

To perform a computational fluid dynamics analysis in parallel, the entire problem must be divided up into pieces, which are then assigned to the available processors. For best performance, this decomposition should evenly distribute the work among the processors, and minimize the number of messages which need to be passed between processors. In addition, support mechanisms such as input/output techniques should be developed to take advantage of system features.

Accomplishments

An existing serial implementation of the two-step explicit predictor-corrector MacCormack algorithm for solution of the 3-D Navier-Stokes equations was ported to the Intel Touchstone Delta massively parallel computer. The parallel design features a 2-D pencilled block data decomposition with only nearest neighbor communications. This is well-suited to the two-dimensional mesh architecture of the Delta machine.

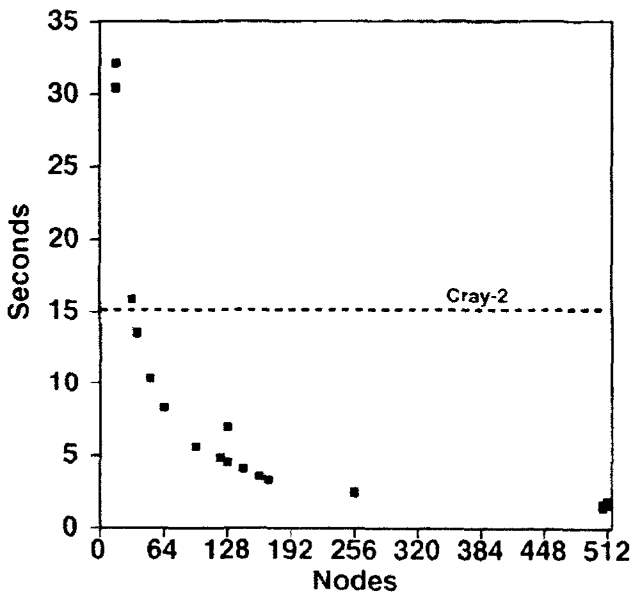
As a test of the implementation, the vortical flow around a 75° swept delta wing was simulated numerically for flow conditions of $\alpha = 30°$, $M_\infty = 1.95$, and a Reynolds number of 4.48×10^6. The computational grid was an $80 \times 80 \times 80$ mesh with an H-H topology. Timing results for the main computational loop indicate a factor of 10 speedup over a single Cray-2 processor when 500+ processors of the Delta machine are utilized (see chart below). The best performance occurs when the work load is evenly balanced across the processors. Various methods for remapping a 13×39 set of blocks into a 16×32 physical processor layout were examined. The best layout occurs by maximizing the number of processors having nearest neighbor communications. This configuration is illustrated in the diagram below.

Future Plans

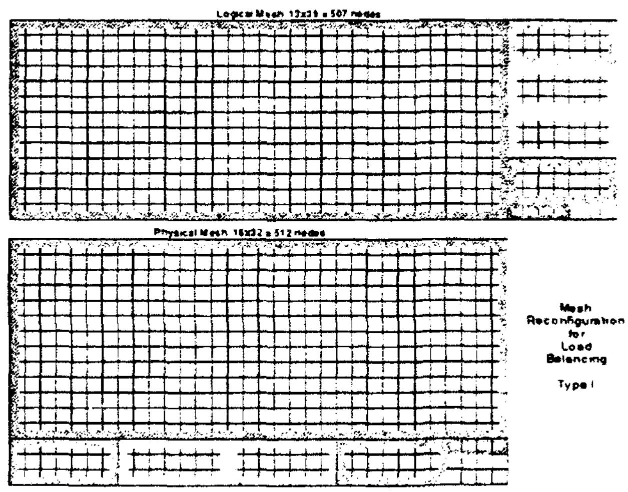
Future work in paralle' ·rocessing for computational fluid dynamics will include implementation of a standard turbulence model under a blocked data decomposition, extension of techniques to other algorithms, such as an explicit-implicit flux-split algorithm, and investigation of methods for parallelizing unstructured grids.

Publications

Scherr, S.J., "Implementation of an Explicit Navier-Stokes Algorithm on a Distributed Memory Parallel Computer," AIAA Paper 93-0063, 1993



Timing results for FDL3D-EP



Reconfiguration for Load Balancing

Advanced Technology Air Vehicle Geometry Data Base

Howard T. Emsley
Interdisciplinary and Applied CFD Section

Research Objective

Accurate geometric representations of USAF aircraft and weapons are necessary to perform many Air Force support activities such as weapon and vehicle modifications, Computational Fluid Dynamic (CFD) analysis, and weapon fit checks. Acquiring necessary geometries is a difficult task due to: a lack of geometric standardization in the industry, the proliferation of computer-aided design tools, and the absence of complete USAF aircraft external geometries. Standardization of geometry formats under the Initial Graphics Exchange Specification (IGES), Product Data Exchange using STEP (PDES), and Computer-aided Acquisition and Logistic Support (CALS) may be the first step toward resolving the problem, but another issue involves the acquisition and storage of air vehicle geometries for use by Air Force organizations. With accurate geometries in hand, the timeliness and ability to perform support activities can be improved, duplication of effort can be reduced, and common geometric configurations can be provided to Air Force research and support organizations.

Approach

To perform support activities for Air Force systems, geometries of USAF aircraft and weapons should be acquired as advanced surface representations and stored in a data base. The goal of this program is to reduce the time required to perform CFD analyses by providing accurate geometric representations to CFD users.

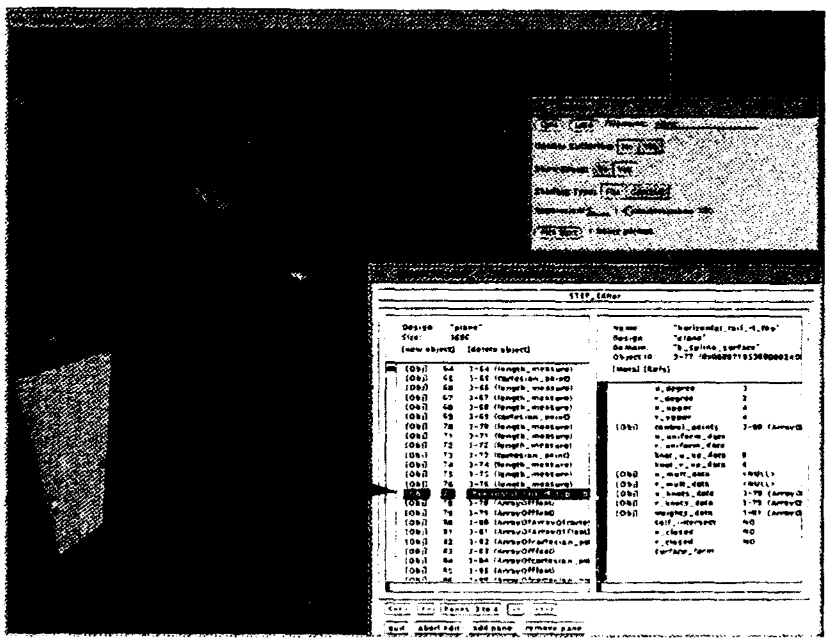
To reach our goal, a data base structure, a set of geometry formats, and the tools to store, retrieve and interactively view complete air vehicle geometries are required. Concurrent with the development of the system, WL is collecting current and future USAF air vehicle geometries to store in the data base. Data formats specified by CALS will be used to acquire the geometries, and MIL standard modifications or clarifications will be addressed to reduce problems in translating geometric representations from different systems.

Accomplishments

A contract to develop the data base structure and interface software was issued to Rensselaer Design Research Center in May 1992. A non-uniform rational b-spline geometry format (STEP NURBS) was selected and the preliminary interface software has been developed. Additional work in progress involves interfacing with existing standards, and the inclusion of topology between geometric entities. The former involves format conversion of IGES entities into STEP NURBS entities for storage and retrieval.

Future Plans

As the contracted effort continues, air vehicle geometries will be collected for conversion and storage in the data base. In-house efforts associated with this effort are concentrating on efficient ways of interfacing NURBS surfaces with existing and developing grid generation tools for CFD.



Advanced Technology Air Vehicle Geometry Data Base Browser and Viewer
(horizontal tail data is displayed in browser and highlighted in viewer)

Computational Electromagnetics

J. S. Shang
Flight Dynamics Directorate

Research Objective

The ultimate objective of the present effort attempts to establish a numerical simulation capability of electromagnetic phenomena for air vehicle technology. The basic process involves the transition of innovative techniques from computational fluid dynamics (CFD) to computational electromagnetics (CEM) for signature analysis and control, target identification, microwave communication, as well as environmental hazard protection. Specific objectives are to first develop efficient numerical procedures for solving time-domain, three-dimensional Maxwell equations, and then map numerical algorithms onto massively parallel processors.

Approach

The time-dependent Maxwell equations for propagating, refracting and diffracting eletromagnetic waves constitutes an initial value problem. The hyperbolic system is solved by characteristic-based schemes, which are derived from eigenvalue and eigenvector analysis. The three-dimensional system of equations, when written in flux vector form, can be diagonalized in each single dimension. The concept of flux-difference splitting will permit windward schemes to mimic the wave mechanism of information propagation. In three-dimensional applications, the cyclic fractional-step or the time-splitting formulation ensures a second-order accurate procedure in time and space. Both finite-difference and finite- volume solving procedures will be implemented on a general curvilinear coordinate system to accommodate complex scatter shapes. Extremely high numerical efficiency for high frequency wave simulation are being achieved through a domain decomposition technique to minimize the data communication on a massively parallel processor.

Accomplishments

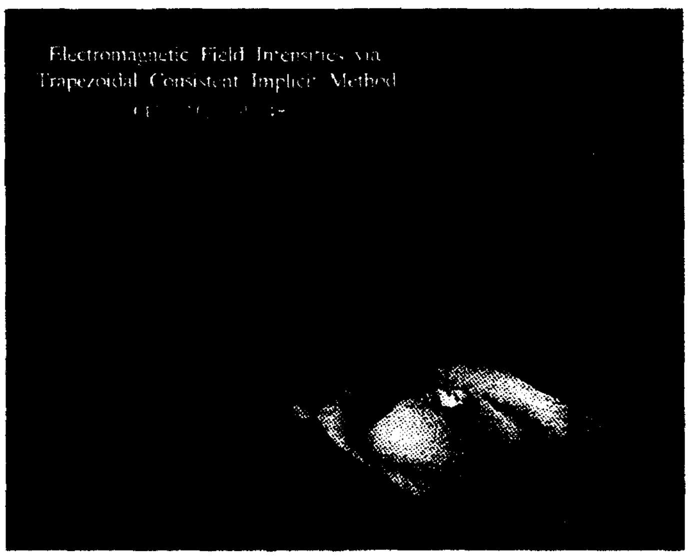
The characteristic-based methods have been validated by applying them to a three-dimensional wave guide and a singular electric dipole problem. The present approach has demonstrated a unique potential in alleviating, and under some special circumstances in eliminating, the fundamental dilemma of reflecting waves from the truncated spatial domain. The numerical procedures also exhibit significantly improved numerical stability properties and reduced alaising errors from the current computational electromagnetic methods in use. A very high data processing rate of six gigaflops has also been attained by a present finite-difference procedure. Most importantly, on a massively parallel system, the present second-order accurate finite-difference procedure is scalable. The scalability is sustained up to 512 nodes, with only a 3 percent parallel efficiency degradation.

Future Plans

Further research will be pursued to refine the finite-volume procedure for an air vehicle radar signature analysis effort. A comparative study is also planned to assess the relative merits of the finite-volume and finite-difference approaches in application to complex scatter geometries, and numerical efficiency on massively parallel computers. The characteristic concept will finally be extended to develop a meshless computing procedure for efficient and accurate CEM applications and diverse dual-use purposes.

Publications

Shang, J. S., "A Fractional-Step Method for Solving 3-D Time Domain Maxwell Equations," AIAA paper 93-0461, Jan. 1993.



Field Intensities for Electric Dipole

List of Authors

The mailing address for all authors is:

WL/FIMC

ATTN: Author's name

2645 Fifth St., Ste. 7

Wright-Patterson AFB, OH 45433-7913

NAME	PHONE NUMBER	E-MAIL ADDRESS
Aftomis, Michael J.	(513) 255-7127	aftomis@fim.wpafb.af.mil
Emsley, Howard T.	(513) 255-4522	emsley@fim.wpafb.af.mil
Fite, Phillip M.	(513) 255-3413	fite@fim.wpafb.af.mil
Gaitonde, Datta	(513) 255-7127	datta@fim.wpafb.af.mil
Gridley, Skip	(513) 255-6207	gridley@fim.wpafb.af.mil
Gordnier, Raymond	(513) 255-7127	gordni@fim.wpafb.af.mil
Josyula, Eswar	(513) 255-2455	josyula@fim.wpafb.af.mil
Kinsey, Don W.	(513) 255-3876	kinsey@fim.wpafb.af.mil
McClure, Thomas H.	(513) 255-4522	mcclure@fim.wpafb.af.mil
Mrozinski, Denis P.	(513) 255-3788	morin@fim.wpafb.af.mil
Mundy, James A.	(513) 255-7127	mundy@fim.wpafb.af.mil
Nelson, John	(513) 255-3413	nelson@fim.wpafb.af.mil
Rizzetta, Donald P.	(513) 255-7127	rizzetta@fim.wpafb.af.mil
Scherr, Stephen	(513) 255-2455	scherr@fim.wpafb.af.mil
Seo, John S.	(513) 255-4522	seo@fim.wpafb.af.mil
Shang, Joseph S.	(513) 255-5616	shang@fim.wpafb.af.mil
Stecklein, Greg	(513) 255-3761	steck@fim.wpafb.af.mil
Strang, William	(513) 255-4522	strang@fim.wpafb.af.mil
Tavares, T. Sean	(513) 255-7127	tavares@fim.wpafb.af.mil
Visbal, Miguel	(513) 255-2455	visbal@fim.wpafb.af.mil
Witzeman, Frank	(513) 255-3788	witzeman@fim.wpafb.af.mil
Wurtzler, Kenneth E.	(513) 255-3761	wurtzler@fim.wpafb.af.mil

DSN phone number is 785 - plus last 4 digits listed.